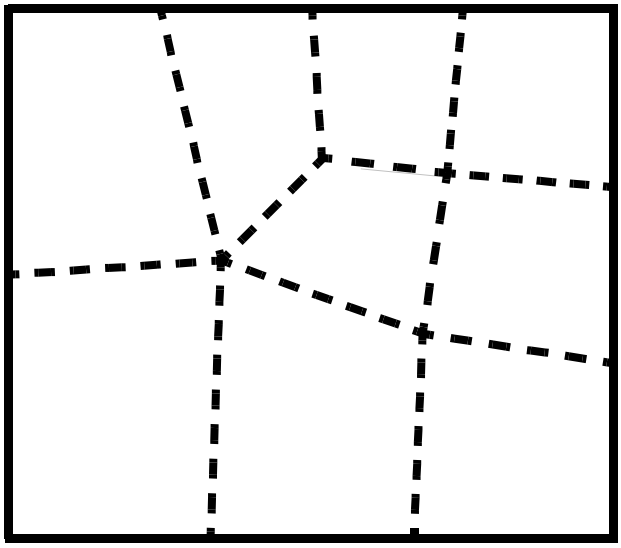
SESAM
USER MANUAL

Prefem



Preprocessor for Generation of
Finite Element Models

DET NORSKE VERITAS

SESAM
User Manual

Prefem

Preprocessor for Generation of
Finite Element Models

June 1st, 2003

Valid from program version 7.1

Developed and marketed by
DET NORSKE VERITAS

DNV Software Report No.: 95-7014 / Revision 3, June 1st, 2003

Copyright © 2003 Det Norske Veritas

All rights reserved. No part of this book may be reproduced, in any form or by any means, without permission in writing from the publisher.

Published by:

Det Norske Veritas
Veritasveien 1
N-1322 Høvik
Norway

Telephone: +47 67 57 99 00
Facsimile: +47 67 57 72 72
E-mail, sales: software.sesam@dnv.com
E-mail, support: software.support@dnv.com
Website: www.dnv.com

If any person suffers loss or damage which is proved to have been caused by any negligent act or omission of Det Norske Veritas, then Det Norske Veritas shall pay compensation to such person for his proved direct loss or damage. However, the compensation shall not exceed an amount equal to ten times the fee charged for the service in question, provided that the maximum compensation shall never exceed USD 2 millions. In this provision “Det Norske Veritas” shall mean the Foundation Det Norske Veritas as well as all its subsidiaries, directors, officers, employees, agents and any other acting on behalf of Det Norske Veritas.

Table of Contents

1 INTRODUCTION 1-1

1.1 Prefem — General Preprocessor for Finite Element Modelling 1-1

1.2 Prefem in the SESAM System 1-3

1.3 How to read the Manual 1-5

1.4 Status List 1-5

2 FEATURES OF PREFEM 2-1

2.1 Modelling Principles 2-1

2.2 Geometry Modelling 2-3

2.3 FE Model Creation 2-4

2.4 Shapes: Modelling Tools 2-4

2.5 Element Library 2-4

2.6 Property Definition 2-8

2.7 Constraints on Geometry and FE Model 2-8
- 2.7.1 Constraints on Geometry 2-8
- 2.7.2 Constraints on Deleting Geometry 2-8
- 2.7.3 Constraints on FE Mesh 2-8
- 2.7.4 Constraints on Element Loading 2-9
- 2.7.5 Constraints on Names 2-11

2.8 Mass Modelling 2-11

2.9 Other Properties 2-12

2.10 Transformations and Coordinate Systems 2-12

2.11 Auxiliary Features 2-13

2.12 Short Description of Commands 2-13

2.13 Transfer of the FE Model through the Input Interface File 2-15
- 2.13.1 Writing and Optimising the Input Interface File 2-15

2.14 Interaction with other SESAM Programs 2-16

3 USER'S GUIDE TO PREFEM 3-1

3.1 Getting Started — the Graphical User Interface 3-2

3.2 Tutorial in Midship Section Modelling 3-7

3.3 Geometry Modelling 3-15
- 3.3.1 Defining Geometry 3-16
- 3.3.2 Defining Shapes 3-23
- 3.3.3 Copying Geometry 3-24
- 3.3.4 Cutting Geometry 3-25
- 3.3.5 Joining Geometry 3-26
- 3.3.6 Rounding off Corners and Cutting Holes 3-26
- 3.3.7 Generating Geometry 3-27
- 3.3.8 Extruding Geometry 3-33
- 3.3.9 The GENERATE Command versus the EXTRUDE Command 3-34
- 3.3.10 Importing DXF File 3-34
- 3.3.11 Geometry Names 3-34

3.4 Creating the FE Model 3-35
- 3.4.1 Controlling the Mesh Creation 3-36
- 3.4.2 Elements for Points 3-38
- 3.4.3 1-D Elements for Lines/Curves 3-38
- 3.4.4 2-D Elements for Surfaces 3-39
- 3.4.5 2-D Elements for Curved Surfaces 3-44
- 3.4.6 3-D Elements for Bodies 3-46
- 3.4.7 Changing the Mesh Created 3-46

3.5 Defining and Assigning (Connecting) Properties 3-46
- 3.5.1 Beam Cross Section 3-47
- 3.5.2 Local Coordinate System for Beam 3-47
- 3.5.3 Eccentricity for Beam 3-48
- 3.5.4 Thickness 3-49
- 3.5.5 Local Coordinate System for Surface Elements 3-49
- 3.5.6 Material 3-49
- 3.5.7 Boundary Condition 3-50
- 3.5.8 Linear Dependency 3-51
- 3.5.9 Point Mass 3-52
- 3.5.10 Numeric Value Input 3-52

3.6 Defining Loads 3-52

3.7 Varying Value Input by Functions 3-55
- 3.7.1 Evaluation of Functions 3-59

3.8 Parameters 3-60

3.9 Selecting Geometry, using Wild-Cards and defining Sets 3-60
- 3.9.1 Graphical Selection of Geometry 3-60
- 3.9.2 Line-Mode Command Selection of Geometry 3-61
- 3.9.3 Using Wild-Cards for Selecting Geometry 3-62

3.9.4 Defining and using Sets for Selection of Geometry ... 3-63

3.10 Defining 'Special' Element Types ... 3-63
3.10.1 Spring, Damper and Mass Elements ... 3-64
3.10.2 Sandwich Elements and Layered Elements ... 3-64
3.10.3 Axi-symmetric Elements ... 3-66

3.11 Transformations and Coordinate Systems ... 3-67
3.11.1 Transformations ... 3-67
3.11.2 Coordinate Systems ... 3-68

3.12 Verifying and Checking the Model ... 3-69
3.12.1 Display and Plot Features ... 3-69
3.12.2 Checking the FE Mesh ... 3-73
3.12.3 Modelling Considerations ... 3-74

4 EXECUTION OF PREFEM ... 4-1

4.1 Program Environment ... 4-1
4.1.1 Starting Prefem from Manager ... 4-1
4.1.2 Reading a Command Input File into Prefem ... 4-2
4.1.3 Starting Prefem as an Individual Program on Unix ... 4-3
4.1.4 Line-Mode Input of Commands and Arguments ... 4-3
4.1.5 Files used by Prefem ... 4-4
4.1.6 Creating Plots for Reports ... 4-6
4.1.7 Background Execution ... 4-6
4.1.8 Command Line Arguments ... 4-7

4.2 Program Requirements ... 4-9
4.2.1 Execution Time ... 4-9
4.2.2 Storage Space ... 4-9

4.3 Program Limitations ... 4-9

5 COMMAND DESCRIPTION ... 5-1

5.1 Selecting Geometry ... 5-2

5.2 Selecting Nodes and Elements ... 5-4

5.3 Functions ... 5-6
LINEAR-2POINTS-VARYING ... 5-10
LINEAR-3POINTS-VARYING ... 5-11
LINEAR-RADIUS-VARYING ... 5-12
CYLINDRICAL-ANGLE-VARYING ... 5-13
CYLINDRICAL-RADIUS-VARYING ... 5-14
VALUE-BETWEEN ... 5-15
ONLY-BETWEEN ... 5-17
SIN / COSIN ... 5-18
EXP / LN ... 5-19

ABS / SIGN / MAX / MIN / DIM / SQRT 5-20

5.4 Detailed Description of Commands 5-21
ADD-DISPLAY 5-23
CHANGE 5-25
CHANGE ARC / INTERSECTION / LINE / SPLINE / NODE-LINE 5-27
CHANGE ELEMENT-ATTRIBUTE 5-29
CHANGE MESH 5-31
CHANGE NAME 5-33
CHANGE NODE 5-34
CHANGE NORMAL-OF-SURFACE / ROTATION-OF-SURFACE 5-35
CHANGE POINT 5-37
CHANGE PROPERTY LOAD load-case TO-MASS 5-38
CHECK 5-40
CHECK CLUSTERED-NODES 5-41
CHECK CLUSTERED-POINTS 5-42
CHECK ELEMENT-SHAPE 5-43
CHECK MESH-TOPOLOGY 5-45
CHECK NON-REGULAR-NODES 5-46
CONNECT 5-47
COPY 5-48
CREATE 5-51
CREATE DESCRIPTION 5-52
CREATE MESH 5-53
CUT 5-54
DEFINE 5-56
DEFINE ARC 5-58
DEFINE BODY 5-59
DEFINE COORDINATE-SYSTEM 5-60
DEFINE DAMPER 5-62
DEFINE INTERSECTION 5-63
DEFINE LAYERED 5-64
DEFINE LINE 5-68
DEFINE MASS-ELEMENT 5-69
DEFINE NODE-LINE 5-70
DEFINE PARAMETER 5-71
DEFINE POINT 5-72
DEFINE POINT name < *update-coordinates* > 5-73
DEFINE PRISM 5-76
DEFINE ROUNDED-CORNER 5-78
DEFINE SECTOR-CORNER 5-79
DEFINE SET 5-81
DEFINE SHAPE 5-83
DEFINE SPLINE 5-85
DEFINE SPRING 5-86
DEFINE SURFACE 5-87
DEFINE TRANSFORMATION 5-90
DELETE 5-92

DELETE MESH 5-94
DELETE PROPERTY 5-95
DISPLAY 5-96
EXIT 5-98
EXTRUDE 5-99
GENERATE 5-102
HELP 5-105
JOIN 5-106
LABEL 5-107
LABEL COLOUR-IDENTIFICATION 5-110
LABEL NODE-SYMBOL 5-111
LOCATE 5-112
MESH 5-113
PLOT 5-115
PRINT 5-117
PROPERTY 5-126
PROPERTY BOUNDARY-CONDITION 5-128
PROPERTY CHANGE 5-130
PROPERTY ECCENTRICITY-BEAM 5-131
PROPERTY HINGE 5-133
PROPERTY INITIAL-DISPLACEMENT 5-135
PROPERTY INITIAL-VELOCITY 5-136
PROPERTY LINEAR-DEPENDENCY 5-137
PROPERTY LINEAR-DEPENDENCY GENERAL-NODE-DEPENDENCY 5-138
PROPERTY LINEAR-DEPENDENCY LINE-LINE-DEPENDENCY 5-139
PROPERTY LINEAR-DEPENDENCY RIGID-BODY-DEPENDENCY 5-141
PROPERTY LINEAR-DEPENDENCY TWO-NODE-DEPENDENCY 5-142
PROPERTY LINEAR-DEPENDENCY TWO-POINT-DEPENDENCY 5-143
PROPERTY LOAD 5-144
PROPERTY LOAD load-case BEAM-CONCENTRATED 5-146
PROPERTY LOAD load-case COMPONENT-PRESSURE 5-148
PROPERTY LOAD load-case CONCENTRATED 5-150
PROPERTY LOAD load-case GRAVITY 5-152
PROPERTY LOAD load-case HYDRO-PRESSURE 5-153
PROPERTY LOAD load-case LINE-LOAD 5-155
PROPERTY LOAD load-case LINE-MOMENT 5-158
PROPERTY LOAD load-case NORMAL-PRESSURE 5-159
PROPERTY LOAD load-case PART-LINE 5-160
PROPERTY LOAD load-case PRESCRIBED-ACCELERATION 5-163
PROPERTY LOAD load-case PRESCRIBED-DISPLACEMENT 5-165
PROPERTY LOAD load-case RIGID-BODY-ACCELERATION 5-166
PROPERTY LOAD load-case RIGID-BODY-VELOCITY 5-167
PROPERTY LOAD load-case TEMPERATURE 5-168
PROPERTY LOCAL-COORDINATE-BEAM 5-170
PROPERTY LOCAL-COORDINATE-SURFACE 5-173
PROPERTY MATERIAL 5-175
PROPERTY MATERIAL material-name ANISOTROPIC 5-176

PROPERTY MATERIAL material-name ANISOTROPIC 2D-ELEMENT 5-177
PROPERTY MATERIAL material-name ANISOTROPIC 3D-SHELL-ELEMENT 5-179
PROPERTY MATERIAL material-name ANISOTROPIC SOLID-ELEMENT 5-182
PROPERTY MATERIAL material-name DAMPER 5-184
PROPERTY MATERIAL material-name ELASTIC 5-185
PROPERTY MATERIAL material-name MASS 5-186
PROPERTY MATERIAL material-name SPRING 5-187
PROPERTY POINT-MASS 5-188
PROPERTY SECTION 5-189
PROPERTY SECTION section-name BAR 5-191
PROPERTY SECTION section-name BOX 5-192
PROPERTY SECTION section-name CHANNEL 5-193
PROPERTY SECTION section-name DOUBLE-BOTTOM 5-194
PROPERTY SECTION section-name GENERAL 5-196
PROPERTY SECTION section-name I 5-198
PROPERTY SECTION section-name L 5-199
PROPERTY SECTION section-name PIPE 5-200
PROPERTY SECTION section-name UNSYMMETRICAL-I 5-201
PROPERTY THICKNESS 5-203
PROPERTY TRANSFORMATION 5-204
RE-COMPUTE 5-206
RE-DISPLAY 5-207
READ 5-208
ROTATE 5-211
SET 5-212
SET COMMAND-INPUT-FILE 5-215
SET DEFAULT 5-216
SET ELEMENT-LENGTH-RATIO 5-220
SET ELEMENT-TYPE 5-222
SET GRAPHICS 5-223
SET GRAPHICS COLOUR 5-227
SET GRAPHICS NUMERICAL-VALUES 5-229
SET GRAPHICS PRESENTATION 5-230
SET GRAPHICS SIZE-SYMBOLS 5-234
SET INSIDE / OUTSIDE 5-236
SET JOURNALLING 5-238
SET MAX-ELEMENT-LENGTH 5-239
SET MESH 5-240
SET MESH-CORNER / NOT-MESH-CORNER 5-243
SET MESH-PARAMETERS 5-244
SET NAMING 5-246
SET NOT-MESH-CORNER 5-248
SET NUMBEROF-ELEMENTS 5-249
SET OUTSIDE 5-250
SET PLOT 5-251
SET PREFIX-NAME 5-253
SET PRINT 5-254

SET PROJECTION 5-257
SET TASK 5-258
SET TOLERANCE 5-260
SET WRITE-MODE 5-261
WRITE 5-262
ZOOM 5-263
..... 5-264

APPENDIX A TUTORIAL EXAMPLES A-1

A 1 Midship Section A-1

A 2 Cylindrical Tank with Flat Bottom and Spherical Top A-4

APPENDIX B THEORY B-1

B 1 Formulae for Sectional Parameters B-1
B 1.1 Bar section B-3
B 1.1.1 Sectional Dimensions B-3
B 1.1.2 Sectional Parameters Computed B-3
B 1.2 Box section B-5
B 1.2.1 Sectional Dimensions B-5
B 1.2.2 Sectional Parameters Computed B-5
B 1.3 Channel section B-7
B 1.3.1 Sectional Dimensions B-7
B 1.3.2 Sectional Parameters Computed B-7
B 1.4 Double-bottom section B-9
B 1.4.1 Sectional Dimensions B-9
B 1.4.2 Sectional Parameters Computed B-9
B 1.5 I (or H) section B-10
B 1.5.1 Sectional Dimensions B-10
B 1.5.2 Sectional Parameters Computed B-10
B 1.6 L section B-12
B 1.6.1 Sectional Dimensions B-12
B 1.6.2 Sectional Parameters Computed B-12
B 1.7 Pipe section B-15
B 1.7.1 Sectional Dimensions B-15
B 1.7.2 Sectional Parameters Computed B-15
B 1.8 Un-symmetrical I section B-17
B 1.8.1 Sectional Dimensions B-17
B 1.8.2 Sectional Parameters Computed B-17

B 2 Units B-20
B 2.1 Example B-20
B 2.2 Consistent Sets of Units B-21

REFERENCES REFERENCES-1

1 INTRODUCTION

1.1 Prefem — General Preprocessor for Finite Element Modelling

Prefem is SESAM's preprocessor for general finite element (FE) modelling. The models may be comprised of truss, beam, membrane, shell and solid elements. More specialised elements like spring, damper and mass elements, sandwich elements, contact elements and axi-symmetric elements are also available.

A model to be used for structural analysis is termed herein a *FE model* whereas a model to be used for hydrodynamic analysis is termed a *panel model*. A panel model may in certain cases be equal to a FE model (the same model is used for both structural and hydrodynamic analysis) but normally it is somewhat different. Modelling a panel model will, however, in principle be the same as modelling a FE model. References in this manual to the terms 'FE model' and 'FE modelling' should therefore also be understood as 'panel model' and 'panel modelling'.

Prefem is characterised by:

- Easy interactive input combined with graphical and printed feedback for model verification
- Extensive data generation features
- A data management system allowing arbitrarily large models

The modelling procedure consists of the following basic steps:

- Model the *geometry* of the structure. The geometry consists of points, lines and curves, surfaces and bodies.
- Give data determining the FE model (element density and types, etc.).
- The *FE model* consisting of elements joined together in nodes is automatically created.
- Assign *properties* such as material data, boundary conditions, loads, etc. to the geometry. These data are automatically transferred to the FE model.

Input is interactively entered in Prefem. The user is guided by prompts for data and graphics functions are available for visualising model data. Data entered are logged on a command log (journal) file (commands not changing the model, for example a display command, are by default not logged). The log file can be used in a new Prefem session to regenerate the model. A standard text editor can be used to modify the log file for the purpose of creating a modified model. The log file is also a documentation and a back-up of the modelling work.

Alternatively to interactive use Prefem can be run in batch mode as explained in Chapter 4. For comprehensive modelling work you may find that editing an input file (which initially may have been a log file) is an efficient and complementary way of working to running Prefem in interactive mode.

Prefem either creates a complete FE model or a first level superelement constituting a part of the complete model. The difference between the two — as seen from Prefem — is that there are some supernodes (or super degrees of freedom) defined for the latter. The term ‘superelement’ is, nevertheless, also used for a complete model made by Prefem, i.e. a first level superelement with no supernodes (or super degrees of freedom).

Figure 1.1 shows an example of a FE model that can be modelled by Prefem.

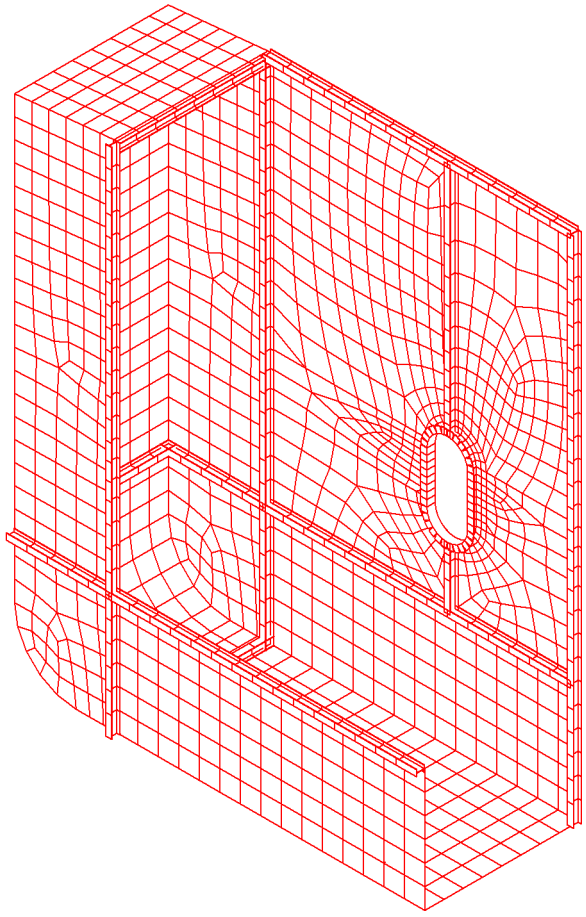


Figure 1.1 FE model with plate elements plus beam elements for stiffeners

1.2 Prefem in the SESAM System

Additionally to Prefem, SESAM comprises a set of preprocessors that are dedicated to various modelling purposes. SESAM's preprocessors are:

Preframe	Modelling superelements consisting of beam, truss and cable elements
Patran-Pre	Modelling superelements of arbitrary shape and complexity consisting of beam, membrane, shell and solid elements
Prefem	Modelling superelements consisting of beam, membrane, shell and solid elements
Presel	Assembling superelements to form the complete model

In addition to these preprocessors SESAM is comprised of a set of hydrodynamic analysis programs, a set of structural analysis programs and a set of postprocessors. The SESAM system overview, an overview of all major SESAM programs and how they communicate, is shown in Figure 1.2.

The program Manager manages an analysis job including modelling, analysis and results processing by activating the proper programs and handling the files involved.

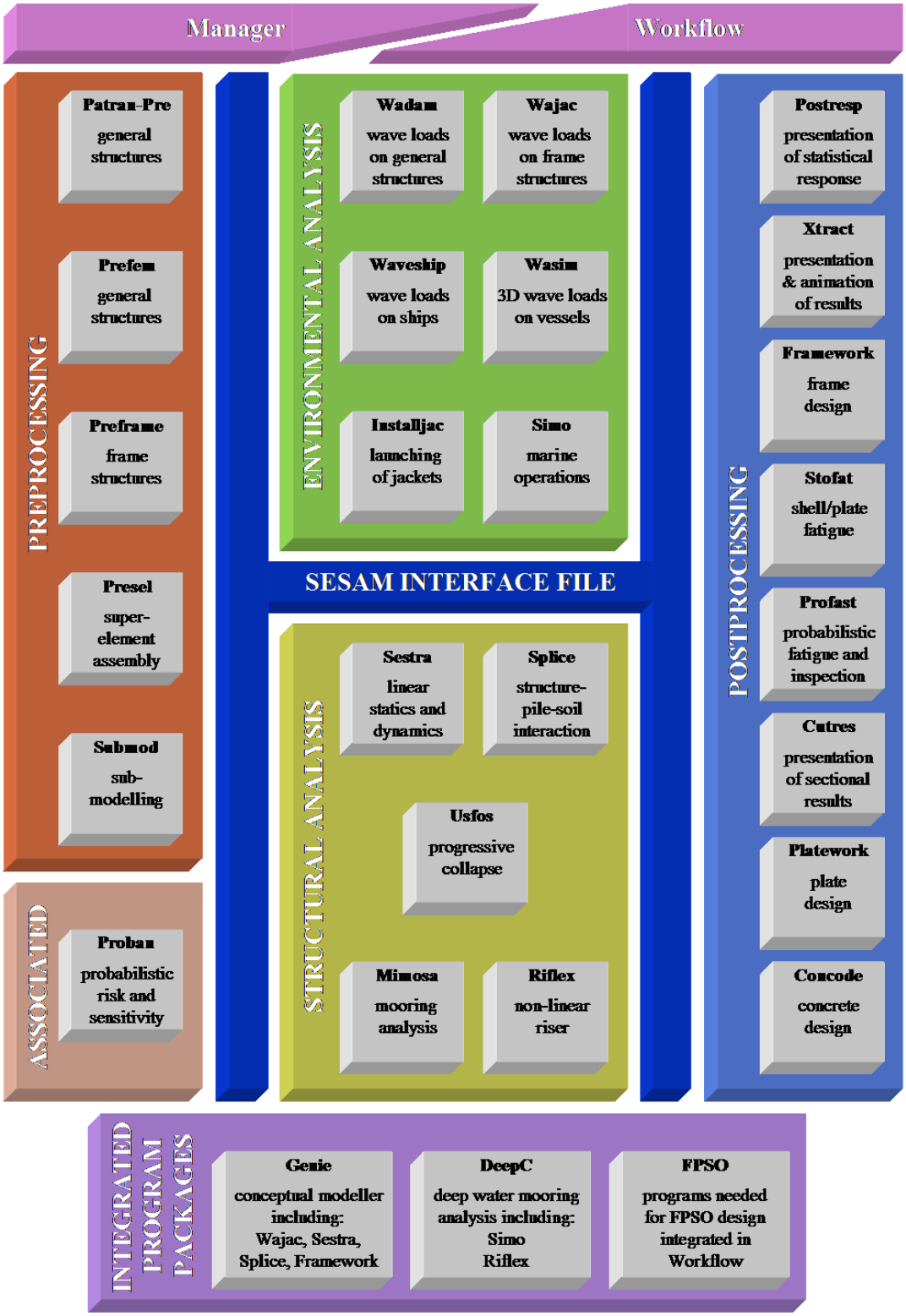


Figure 1.2 SESAM overview

1.3 How to read the Manual

If you are a new user you may proceed as follows:

- Read Section 2.1 Modelling Principles to learn about some basic concepts of Prefem.
- Read Section 2.2 Geometry Modelling and Section 2.3 FE Model Creation for introductions to modelling.
- Go then to Section 3.1 Getting Started — the Graphical User Interface to learn how to operate Prefem and to Section 3.2 Tutorial in Midship Section Modelling were you will be guided through a small but complete modelling session.
- After completing the tutorial you may achieve a more complete understanding of geometry modelling, FE model creation, defining and assigning properties as well as defining loads from Section 3.3 through Section 3.6.
- Having done the above steps you should have an adequate understanding of Prefem enabling you to proceed with practical modelling work while referring to the other sections of this manual, in particular the sections of Chapter 3 and the description of all Prefem commands in Chapter 5.

This manual is otherwise organised as follows:

Chapter 2 FEATURES OF PREFEM contains an introductory description of the major features of Prefem.

Chapter 3 USER'S GUIDE TO PREFEM explains how to go about creating a complete model ready for analysis. All major features and several minor features are described. The chapter does not contain a full description of all program features, though; a complete understanding of all features of Prefem can only be obtained through training in use of the program while referring to Chapter 5.

Chapter 4 EXECUTION OF PREFEM contains more special information not intended for the new user who will be using Manager to control his SESAM analysis. The chapter explains how to start Prefem outside Manager and operate it in line-mode (not using the graphical user interface). The files used by Prefem are also explained. Practical information is provided on how to operate Prefem and manipulate its files in various ways. Built-in and hardware dependent requirements and limitations are also described.

Chapter 5 COMMAND DESCRIPTION explains in detail all commands of Prefem. The commands and subcommands are sorted alphabetically.

Appendix A TUTORIAL EXAMPLES contains a couple of examples of use.

Appendix B THEORY contains the formulae employed by the program for computing sectional parameters for the various types of beam cross section. Guidance in how to choose a consistent set of units for your analysis is also found here.

1.4 Status List

There exists for Prefem (as for all other SESAM programs) a Status List providing additional information. This may be:

- Reasons for update (new version)

- New features
- Errors found and corrected
- Etc.

Use the program Status for looking up information in the Status List. Use the command HELP to start Status.

2 FEATURES OF PREFEM

Prefem is a general purpose interactive graphic program for modelling of FE models.

2.1 Modelling Principles

The modelling principles of Prefem are based on a dual model concept; see Figure 2.1:

- A *geometry model* consisting of the entities points, lines and curves, surfaces and bodies. The *user* defines this model.
- A *FE model* consisting of elements joined in nodes. The *program* automatically creates this model based on the geometry model and data determining the FE mesh.

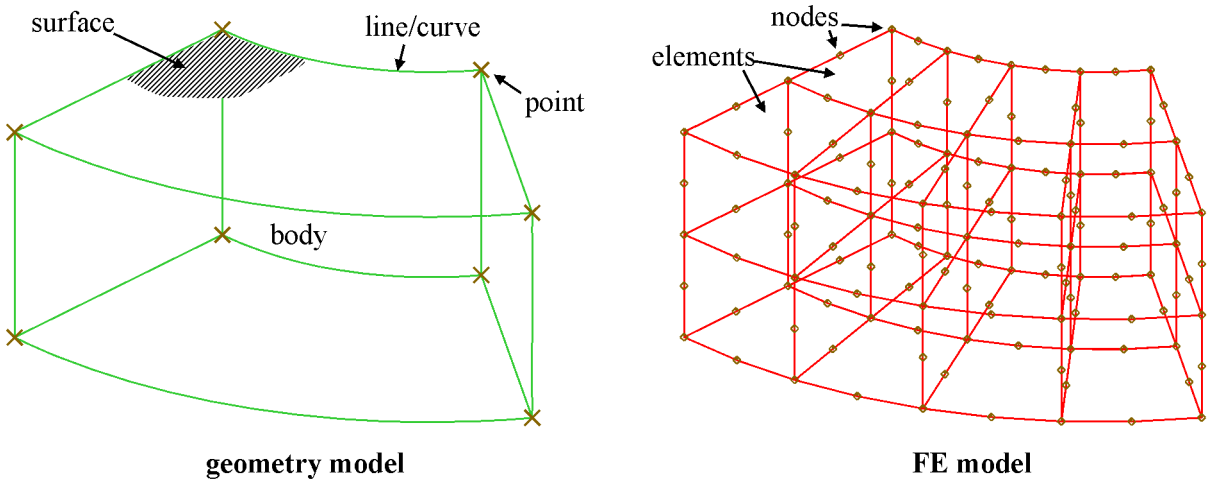
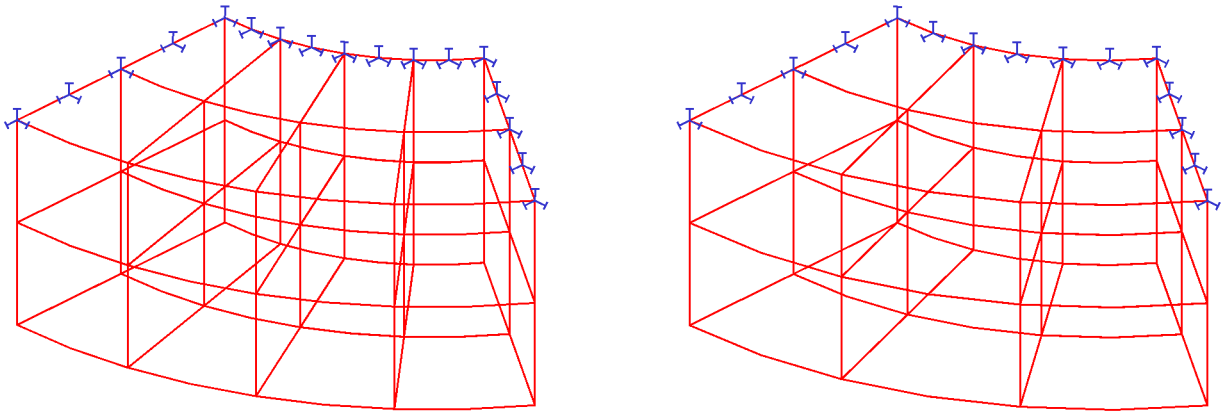
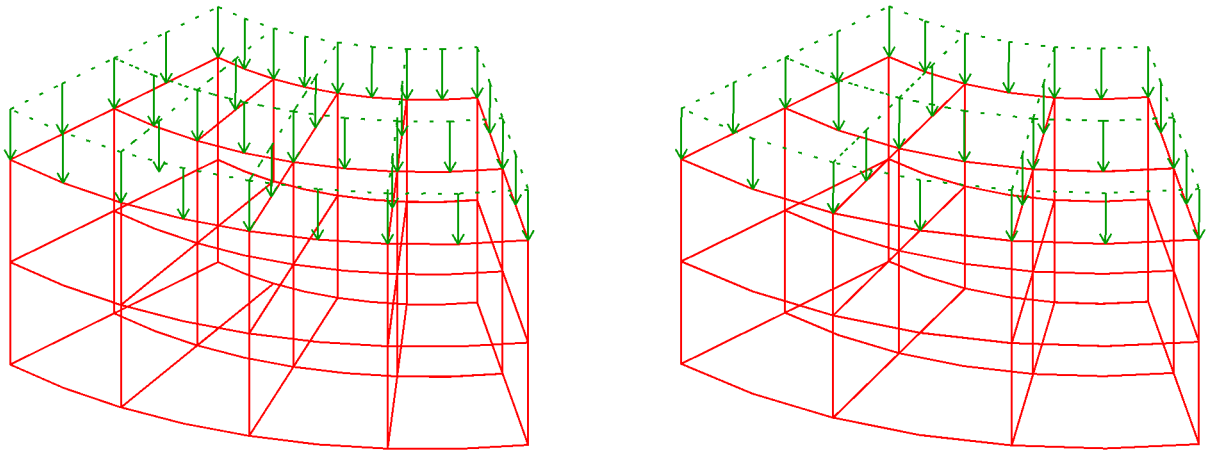


Figure 2.1 Dual model concept: user defines the geometry model, program creates the FE model

Properties such as material data, boundary conditions, loads, etc. are assigned or connected to the geometry and automatically transferred to the FE model. This simplifies the input of properties. It also allows the user to change the FE mesh, e.g. increase or decrease the element density, without having to re-enter the properties; see Figure 2.2. The geometry can also be changed without re-defining the properties. If, for example, a surface with pressure load is deleted, the load will automatically disappear. On the other hand, if only the shape of the surface is changed, the load will remain.



Boundary conditions defined, then mesh deleted and new mesh created, boundary conditions prevail



Load (normal pressure) created, then mesh deleted and new mesh created, load prevails

Figure 2.2 Properties assigned to geometry model allows changing mesh without re-entering data

Throughout the modelling the user will, therefore, basically only work with the geometry: define the geometry, delete and change it, refer to it when defining properties and display it. The FE model, being the objective of the modelling, is a concern of the program: the user will basically never have to refer directly to it (he will, however, display the FE model for verification purposes). This approach is logical as it brings the modelling work close to the definition of the structure, i.e. the drawings or a CAD model. The approach is also efficient as the time consuming task of defining elements and nodal coordinates is left to the program. Efficiency is also gained as quick and easy re-meshing of the model is facilitated by the fact that properties are assigned to the geometry instead of to the FE model.

Modelling with Prefem will typically consist of the following steps:

1 Defining the geometry model

2 Defining data determining the FE mesh (element density and types, etc.) and requesting automatic creation of the FE mesh (e.g. by the command MESH ALL)

3 Defining properties (material data, boundary conditions, loads, etc.)

4 Verifying the model by printing and displaying data

5 Storing the completed FE model on the Input Interface File (see Section 2.13) for transfer to analysis programs

Step 3, defining properties, may as well be performed prior to or in combination with step 2.

In addition to creating FE models for structural analysis Prefem is used for creating hydrodynamic analysis models, so-called panel models (analysed by Wadam). The steps of modelling a panel model are the same as for modelling a FE model.

When exiting the program during modelling Prefem will automatically save the model thereby allowing the modelling work to be resumed at a later stage.

2.2 Geometry Modelling

The geometrical entities are described in Table 2.1. The complete geometry model may consist of any number and any combination of these entities.

Table 2.1 Geometrical entities

Entity		**Definition**
Point		Coordinates (x, y, z)
Line/ curve	Line	Straight line between two points
	Arc	Circular arc defined by two points and a centre point
	Intersection	Intersection curve between two shapes and going from one point to another, a third point is required for picking the proper intersection segment
	Spline	B-spline curve defined by an arbitrary number of points
	Node-line	Straight line segments between an arbitrary number of points (and with nodes in the points when a FE mesh is created)
	Curve	Curve made by the GENERATE command using cylindrical or spherical coordinate system
Surface		Enclosed by an arbitrary number of lines, arcs, splines and node lines
Body		Enclosed by top and bottom surfaces and an arbitrary number of side surfaces

2.3 FE Model Creation

Having defined the geometry consisting of points, lines and curves, surfaces and bodies the following data must be defined to enable automatic creation of the FE model:

- Data determining the element discretisation, i.e. number and spacing of elements for lines and curves (the number of elements is given when defining the lines/curves, but may be changed later)
- Desired type(s) of element

The FE model (mesh) is automatically created by the command MESH as illustrated in Figure 3.29. A mesh consists of elements joined together in nodes.

2.4 Shapes: Modelling Tools

Shapes are tools in the form of surfaces for defining geometry and for projecting FE mesh onto. In itself a shape does not constitute a part of neither the geometry model nor the FE model. The following shapes are available and defined by the DEFINE command:

- Plane
- Cylinder
- Cone
- Sphere
- Interpolation surface between two unconnected sets of lines/curves

Shapes are used for the following purposes:

- When defining a surface thereby ensuring that the FE mesh to be created for the surface will be projected onto the shape (see command DEFINE SURFACE)
- For the same purpose as above but done *after* defining a surface (see command SET PROJECTION)
- For defining points by intersecting three shapes (see command DEFINE POINT name < SHAPE-INTERSECTION)
- For defining curves by intersecting two shapes (see command DEFINE INTERSECTION)

2.5 Element Library

The element types that may be created by Prefem are presented in table Table 2.2 and illustrated in Figure 2.3. Some of the elements are not available in Sestra, SESAM's linear analysis program. Such elements are

for use in SESAM's non-linear analysis program Advance. Names of such Advance-only elements are given in parentheses.

Table 2.2 Element types of Prefem

Element type **Name used in command**	**Short name**	**Description**	nodes	d.o.f. per node
TRUSS	TESS	Truss	2	3
BEAM-2NODES	BEAS	Beam	2	6
BEAM-3NODES	SECB	Curved beam	3	6
AXISYMMETRIC-3+3NODE-CONTACT	CTAQ	Axi-symmetric contact element, 3 nodes per side (INTER3A) *	6	2
MEMBRANE-3NODES	CSTA	Constant strain triangle	3	3
MEMBRANE-4NODES	LQUA	Linear quadrilateral	4	3
MEMBRANE-6NODES	ILST	Isoparametric linear strain triangle	6	3
MEMBRANE-8NODES	IQQE	Isoparametric quadratic strain quadrilateral	8	3
AXISYMMETRIC-3NODES	AXCS	Axi-symmetric versions of the corresponding membrane elements	3	2
AXISYMMETRIC-4NODES	AXLQ		4	2
AXISYMMETRIC-6NODES	AXLS		6	2
AXISYMMETRIC-8NODES	AXQQ		8	2
SHELL-3NODES	FTRS	Triangular flat thin shell	3	6
SHELL-4NODES	FQUS	Quadrilateral flat thin shell	4	6
SHELL-DRILLING-3NODES	FTAS	Triangular flat thin shell with drilling degrees of freedom	3	6
SHELL-DRILLING-4NODES	FQAS	Quadrilateral flat thin shell with drilling degrees of freedom	4	6
SHELL-6NODES	SCTS	Subparametric curved triangular thin/thick shell	6	6
SHELL-8NODES	SCQS	Subparametric curved quadrilateral thin/ thick shell	8	6
SHELL-9NODES	HCQS	9 node reduced integration doubly curved shell (S9R5) *	9	6
SANDWICH-6NODES	MCTS	Curved triangular sandwich element based on SCTS	6	6

Table 2.2 Element types of Prefem

Element type **Name used in command**	**Short name**	**Description**	nodes	d.o.f. per node
SANDWICH-8NODES	MCQS	Curved quadrilateral sandwich element based on SCQS	8	6
LAYERED-6NODES	LCTS	Curved triangular layered shell based on SCTS	6	6
LAYERED-8NODES	LCQS	Curved quadrilateral layered shell based on SCQS	8	6
CONTACT-4+4NODES	CTLQ	Contact element, 4 nodes per face (INTER4) *	8	3
CONTACT-8+8NODES	CTCQ	Contact element, 8 nodes per face	16	3
CONTACT-9+9NODES	CTMQ	Contact element, 9 nodes per face (INTER9) *	18	3
SOLID-6NODES	TPRI	Triangular prism	6	3
SOLID-8NODES	LHEX	Linear hexahedron	8	3
SOLID-15NODES	IPRI	Isoparametric triangular prism	15	3
SOLID-20NODES	IHEX	Isoparametric hexahedron	20	3
SOLID-21-TO-27-NODES	GHEX	21 to 27 node full integration brick (C3D27) *	21-27	3
SPRING TO-GROUND	GSPR	Spring-to-ground	1	1 - 6
SPRING AXIAL	AXIS	Axial spring	2	6
DAMPER TO-GROUND	GDAM	Damper-to-ground	1	1 - 6
DAMPER AXIAL	AXDA	Axial damper	2	6
MASS-ELEM ONE-NODE	GMAS	General one node mass element	1	1 - 6
		* Elements only available in Advance		

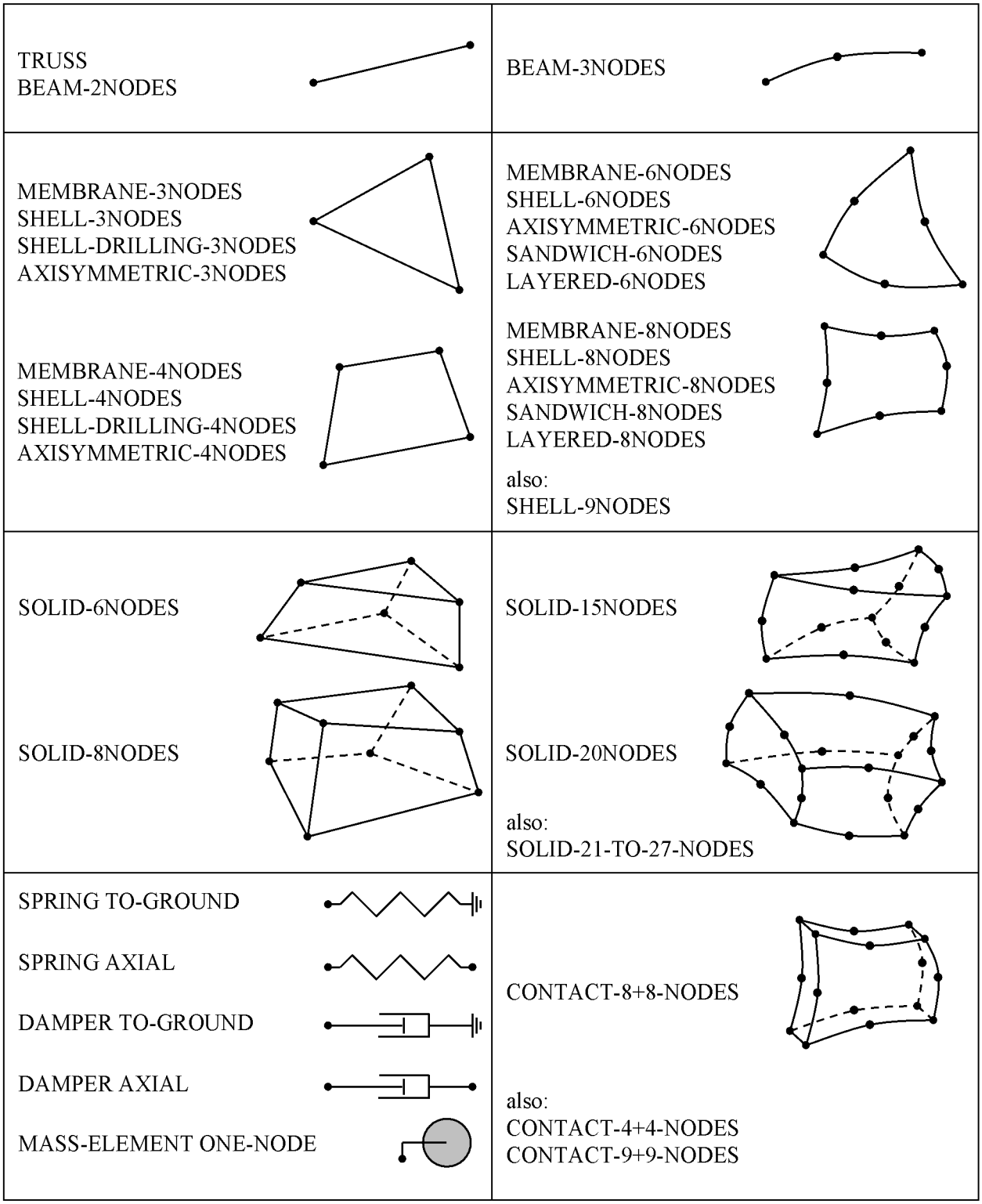


Figure 2.3 Graphical illustration of SESAM's element types

2.6 Property Definition

Material data, beam cross sections, plate thicknesses, boundary conditions, loads, etc. are so-called properties that are assigned or connected to the relevant geometry by referring to the appropriate geometry names. The properties will automatically be transferred to the FE model irrespective of whether the properties are defined before or after the FE mesh creation.

Most properties are defined and assigned to the relevant geometry by one command (PROPERTY). The following properties, however, are first defined (command PROPERTY) and secondly assigned to the relevant geometry (by the CONNECT command):

- Material data
- Beam cross sections
- Layered element data (often used for stiffened plate modelling)

Some properties may, however, also be assigned directly to nodes and elements after creating the FE mesh. Note that these properties will disappear when the mesh is deleted.

2.7 Constraints on Geometry and FE Model

2.7.1 Constraints on Geometry

- Two points are not allowed to have the same coordinates.
- Two lines/curves/surfaces/bodies *are* allowed to have exactly the same definition. This will give double sets of nodes and is normally not the proper way of modelling; see Section 2.3 for notes and Figure 3.32 for an illustration of this.
- A circular arc defined by the:
  — DEFINE ARC command must be less than 180 degrees.
  — DEFINE INTERSECTION command must be less than 360 degrees.

2.7.2 Constraints on Deleting Geometry

- Surfaces cannot be deleted if bodies are defined based on the surfaces.
- Lines/curves cannot be deleted if surfaces are defined based on the lines/curves.
- Points cannot be deleted if lines/curves are defined based on the points.

2.7.3 Constraints on FE Mesh

- Node and element numbers are automatically assigned by Prefem during creation of the FE mesh and cannot be changed by the user (the modelling principles implies that he will have no reason for doing so).

- A maximum of four mesh-corners are allowed for a surface; see Figure 3.31. An exception is when there is only one single element between two mesh-corners; see Figure 3.39.
- If the sum of the numbers of elements for all lines/curves surrounding a surface is an odd number then a triangular element is automatically inserted in the sharpest corner. The user may overrule this by selecting any corner for the triangular element. See the illustration of Figure 2.4.
- There are cases where a FE mesh logically cannot be created, e.g. when many elements are requested for one side or for two adjoining sides compared with the number of elements for other sides of a quadrilateral. A too high degree of mesh refinement is requested. See the illustration of Figure 2.5.
- The solid element mesh for a body must be prismatic, i.e. the discretisation (mesh topology) of the top and bottom surfaces must be equal so that there is no mesh refinement in the top-bottom-surface-direction. This is consistent with the requirement to the geometry.

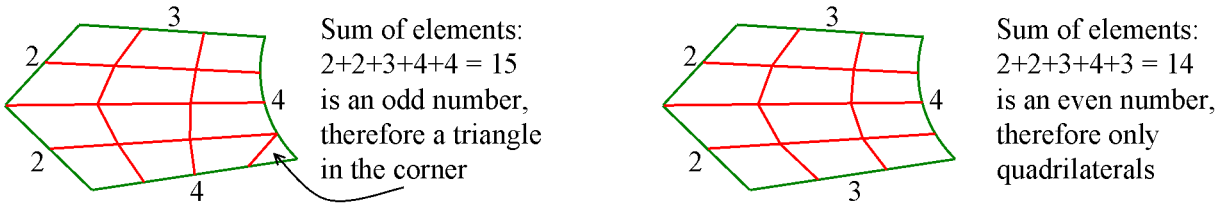


Figure 2.4 Surfaces with odd and even sum of number of elements

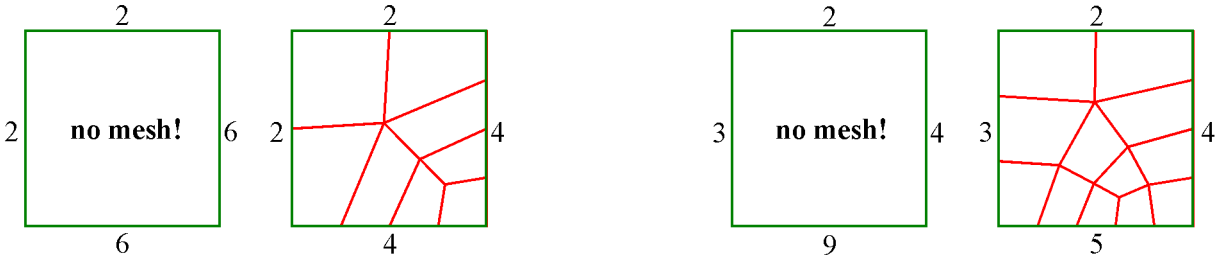


Figure 2.5 Surfaces with impossible and possible mesh refinements

2.7.4 Constraints on Element Loading

Several types of loads may be defined in Prefem: concentrated load, line distributed load, surface normal pressure, etc. Not all types of loads can be applied to all types of elements. Table 2.3 shows the relevant types of loads for the various types of elements of the Sestra analysis program. Refer to the relevant user manual for other analysis programs.

Some of the types of loads created by Prefem are related to the nodes rather than to the elements. These are therefore allowable independently of the type of element used. These elements are:

- Concentrated nodal load (including forces and moments)

- Prescribed nodal displacement
- Prescribed nodal acceleration
- Rigid body acceleration
- Rigid body velocity

Table 2.3 Allowable loads for a Sestra analysis depending on type of element

Type of element		beam concentrated	line load	line moment	part line	normal pressure	component pressure	temperature	gravity (general inertia)	hydro pressure
Truss, beam	Truss							x		
	Beam	x	x		x			x	x	
	Curved beam		x	x	2			x	x	
Membranes	Constant strain triangle		1		2			x	3	
	Linear quadrilateral		1		2			x	3	
	Isoparametric linear strain triangle		1		2			x	3	
	Isoparametric quadratic strain quadrilateral		1		2			x	3	
	Axi-symmetric elements corresponding to membranes							4	4	
Shells	Triangular flat thin shell		x	x	2	x	x	x	x	x
	Quadrilateral flat thin shell		x	x	2	x	x	x	x	x
	Subparametric curved triangular thin/thick shell		x	x	2	x	x	x	x	x
	Subparametric curved quadrilateral thin/thick shell		x	x	2	x	x	x	x	x
	Curved triangular sandwich element based on SCTS		x	x	2	x	x	x	x	
	Curved quadrilateral sandwich element based on SCQS		x	x	2	x	x	x	x	
	Curved triangular layered shell based on SCTS		x	x	2	x	x	x	x	x
	Curved quadrilateral layered shell based on SCQS		x	x	2	x	x	x	x	x

Table 2.3 Allowable loads for a Sestra analysis depending on type of element

Type of element		beam concentrated	line load	line moment	part line	normal pressure	component pressure	temperature	gravity (general inertia)	hydro pressure
Solids	Triangular prism		x			x	x	x	x	x
	Linear hexahedron		x			x	x	x	x	x
	Isoparametric triangular prism		x			x	x	x	x	x
	Isoparametric hexahedron		x			x	x	x	x	x
Spring, damper, mass	Spring-to-ground									
	Axial spring									
	Damper-to-ground									
	Axial damper									
	General one node mass element								x	

Notes in the table above:

1 Only the in-membrane-plane component of a line-load is taken into account in Sestra, see further explanation for the PROPERTY LOAD load-case LINE-LOAD command.

2 The extent of the load is adjusted to match whole elements and the magnitude is scaled to compensate for this adjustment.

3 In spite of note 1 above gravity out-of-membrane-plane is allowed.

4 Only as axi-symmetric load

2.7.5 Constraints on Names

- Names given by the user cannot start with the letter X (this is reserved for program-generated names).
- Length of names is limited to 8 characters.

2.8 Mass Modelling

The Sestra analysis program will automatically generate element mass based on the volume of the elements and density of the material. Additional mass may be modelled in Prefem as follows:

- The PROPERTY POINT-MASS command enables definition of point masses (diagonal mass matrices) for selected geometry points.
- The DEFINE MASS-ELEMENT command enables definition of mass elements (full mass matrices) for selected geometry points.
- The CHANGE PROPERTY (or PROPERTY CHANGE) LOAD load-case TO-MASS command enables conversion of selected load-cases for selected geometry points/lines to node masses. This is relevant for the following load types:
  — CONCENTRATED in general (independent of type of element)
  — BEAM-CONCENTRATED, LINE-LOAD and PART-LINE when applied to 2 node beam elements

2.9 Other Properties

In addition to defining material data, beam cross sections, etc. as described in Section 2.6 the PROPERTY command is used for defining properties like:

- Local coordinate systems for:
  — Beams
  — Surface elements (only relevant for layered elements)
- Eccentricities for beam elements
- Linear dependencies by which one or more degrees of freedom or nodes may be made linearly dependent of one or more other degrees of freedom or nodes
- Initial displacements and initial velocities, this is relevant in connection with dynamic analysis

2.10 Transformations and Coordinate Systems

Transformations and coordinate systems are defined by the DEFINE command for various purposes. Both are first defined and given names and are then available for later reference.

- Transformations composed of translations, rotations, scaling and/or mirroring are used:
  — For defining boundary conditions and loads in a transformed (askew) coordinate system
  — For orientating spring-to-ground and damper-to-ground elements
  — For copying geometry
- Coordinate systems of either cylindrical or spherical type are used:
  — For defining point coordinates
  — By the GENERATE command (note that the GENERATE command will, unless referring to an existing coordinate system, implicitly define a coordinate system)
  — For defining boundary conditions and loads, radial and circumferential boundary conditions are for example easily defined by referring to a cylindrical or spherical coordinate system

2.11 Auxiliary Features

There are some auxiliary features available for making the modelling easier and for verifying the model, these are briefly presented below.

- DEFINE SET may be used to define a named set (selection) of the geometry, this set may then be referred to whenever reference to existing geometry is required.
- DEFINE PARAMETER may be used to give a named parameter a value, this parameter may then be referred to when e.g. defining point coordinates using the *update mode*; see Section 3.3.1.
- The model may be displayed in different ways using the DISPLAY command and plotted (sent to printer or file) using the PLOT command. Other commands that may be used in conjunction with displaying and plotting are: LABEL, ZOOM, ROTATE, LOCATE and SET GRAPHICS. Alternatively to entering these commands there are several so-called 'direct access buttons' and 'shortcut commands' providing quicker access to the same functionality, these are described in Section 3.1.
- The PRINT command is available for printing data on the screen or to a file.
- As an alternative to the PRINT command the 'direct access button' named 'Info' allows quick printing of various information on the geometric model. See the description of the 'direct access buttons' in Section 3.1 on this.
- In addition to influencing the display of a model through SET GRAPHICS the SET command is used for several other purposes:
  - — Deciding type of element to use, the number of elements along the lines and their spacing (element length ratio)
  - — Setting various tolerances used for deciding match/no-match situations, e.g. for coordinates
  - — Setting various control parameters for the mesh creation
  - — Destination and file name for print and plot of data
  - — Determining inside/outside of surfaces in connection with application of loads

2.12 Short Description of Commands

A short description of each main command of Prefem is given below:

ADD-DISPLAY	adds graphical information to the current display, i.e. geometry, FE mesh or loads.
CHANGE	changes data such as geometry, shapes, properties, transformations, one node elements, nodes, etc.
CHECK	checks the shape of elements created and mesh topology (i.e. whether a FE mesh can be created or not).
CONNECT	connects materials previously defined to geometry, cross sections to beams and layered element data to surfaces.
COPY	copies geometry thereby defining more geometry.

CREATE	creates mesh, a command equivalent to MESH.
CUT	cuts lines and surfaces thereby defining new geometry.
DEFINE	defines geometry, shapes, one node elements, layered element data, transformations, coordinate systems, parameters.
DISPLAY	displays geometry and FE mesh.
EXTRUDE	extrudes and copies geometry.
GENERATE	generates geometry, and efficient way of defining regular geometry.
HELP	provides help on how to get support, status of Prefem (the Status List) and commands.
JOIN	joins two bodies into one.
LABEL	labels (tags) information on display: geometry names, symbols for nodes and boundary conditions, node and element numbers, symbols for mesh-corners and element normals, section and material names, etc.
LOCATE	adds dotted lines indicating location of geometry in relation to the displayed mesh.
MESH	creates mesh for the whole geometry or for a selected part of the geometry.
PLOT	creates a plot file of the screen display, alternatively the geometry or FE mesh identified.
PRINT	prints data: geometry, FE mesh, loads, material data, etc.
PROPERTY	defines properties.
RE-COMPUTE	re-computes loads. In certain situations, e.g. when loads previously defined are being changed and a display of the new loads is desired (by ADD-DISPLAY), this command is required. (The command is not required prior to the WRITE command to get the correct loads on the Input Interface File.)
RE-DISPLAY	re-displays geometry or FE mesh to get a display with current setting of labels etc.
READ	reads either an Input Interface File (for displaying only, no modifications may be done) or a DXF file (a geometry model is established based on the information).
ROTATE	rotates the display on the screen.
SET	sets various control and performance parameters.
WRITE	writes the current model onto an Input Interface File.
ZOOM	zooms in and out on the screen display.
#	reads a given number of commands from a command input file into Prefem.

DELETE	deletes geometry, FE mesh, properties, one node elements, transformations.
EXIT	involves exiting from Prefem. The model and log files are closed and saved.

2.13 Transfer of the FE Model through the Input Interface File

As is the case for all SESAM preprocessors, the model created by Prefem is transferred to the hydrodynamic and/or structural analysis programs via the Input Interface File which forms a part of the SESAM Interface File system.

The Input Interface File, the T-file, is a sequential ASCII character file with 80 character long records. The straightforward definition of the file enables external programs to be connected to the SESAM system with comparative ease.

One interface file will be created for each superelement. The name of the file will be:

prefixT#.FEM

where:

- 'prefix' is an optional character string that may and may not include a directory specification, the string is common for all superelements in a superelement model.
- 'T' is a mandatory character identifying this as an Input Interface File, a T-file, as opposed to a Loads Interface File, L-file, which uses character L and a Results Interface File, R-file, which uses character R.
- '#' is the superelement number, the identifier of the superelement.
- 'FEM' is a mandatory file extension.

Normally, the user may find it convenient to leave the prefix void. This is also the default condition.

An example of a name of an Input Interface File is:

ABCT5.FEM

When using the superelement technique, all superelements belonging to the same model should have the same file prefix. If the above file — superelement 5 — is one of several files of a superelement model then all Input Interface Files should be named ABCT#.FEM, where # is the superelement number.

2.13.1 Writing and Optimising the Input Interface File

Whether or not to write the Input Interface File is normally controlled by Manager. If you want to produce the file you should check the appropriate box prior to starting Prefem. The Input Interface File is then automatically written when you exit Prefem using the command EXIT. (This makes the Prefem command WRITE superfluous.)

Note: If you in Windows close the Prefem window by the X in the upper right corner (or by the Close (Alt+F4) command of the window menu) then the Input Interface File will not be written even though you have requested this when starting Prefem. This feature may be used if you change your mind and decide not to write the file after having started Prefem.

Also controlled by Manager is the optimization of the internal node numbering (going from 1 to N, where N is the number of nodes in the model) in order to minimise the bandwidth of the stiffness matrix. When checking the appropriate box prior to starting Prefem the auxiliary program Bpopt is automatically run after exiting Prefem. Bpopt then optimises the internal node numbering and produces a revised Input Interface File.

Note: Unless Sestra's Multifront equation solver is used the optimization should *always* be performed or else the CPU time may be excessively large! The Multifront equation solver is presently available for static analysis only implying that optimization should be performed for all dynamic analyses. Moreover, if the model created is a first level superelement which is to be coupled with other superelements then this optimization should be performed prior to reading the model into Presel.

If you run Prefem in a 'non-standard' way (not using the command Model | General Prefem in Manager) you need to either use a command line argument, see Section 4.1.8, or you need to use the command WRITE in order to produce the Input Interface File, the T-file. In such case you also need to run Bpopt manually after exiting Prefem (i.e. if you need to optimise the internal node numbering).

2.14 Interaction with other SESAM Programs

All model characteristics featured by Prefem are not necessarily accepted by a particular analysis program. The Sestra linear analysis program will accept most model data created by Prefem. When creating a model for the non-linear analysis program Advance you should make sure the model is consistent with the analysis program.

3 USER'S GUIDE TO PREFEM

The purpose of Prefem is to create a FE model for structural analysis or a panel model for hydrodynamic analysis. This is done by first defining a geometry model and assigning element discretisation (FE mesh) data to this model. Based on this information the FE model is created automatically. Secondly, various properties like beam cross sections and eccentricities (offsets), surface thicknesses, material data and boundary conditions are given and assigned to the appropriate parts of the geometry model. Finally, the loads are defined and assigned to the geometry model. All properties and loads are automatically transferred to the FE model. The output from Prefem is the Input Interface File, see Section 2.13, containing the FE model.

This user's guide explains how to:

- Get started using the graphical user interface. See Section 3.1.
- Create a small but complete model, a brief tutorial. See Section 3.2.
- Define the *geometry model*. See Section 3.3.
- Create the *FE model* by determining the element discretisation, selecting element type(s) and requesting automatic creation of the FE mesh. See Section 3.4.
- Define and assign (connect) *properties* (material data, boundary conditions, etc.). See Section 3.5.
- Define *loads*. See Section 3.6.
- Give *varying value (functions) input* for properties (especially loads). See Section 3.7.
- Define and use *parameters*. See Section 3.8.
- Select geometry and using *sets* and *wild-cards* for the same purpose. See Section 3.9.
- Define *'special' element types*: spring, damper, mass, sandwich and layered elements. See Section 3.10.
- Define and use *transformations* and *coordinate systems*. See Section 3.11.
- *Display, verify* and *check* the FE model. See Section 3.12.

3.1 Getting Started — the Graphical User Interface

Prefem is started from the SESAM Manager by clicking **Model | General Prefem**.

See Section 4.1.3 for how to start Prefem outside Manager (Unix only).

The main part of the graphical user interface is the graphic-mode window. The appearance of this window is principally the same on PC (Windows) and Unix. On PC there are also a print window and a message window. Print requested by the user appears in the print window whereas various program messages appear in the message window. Figure 3.1 illustrates the three Prefem windows on a PC.

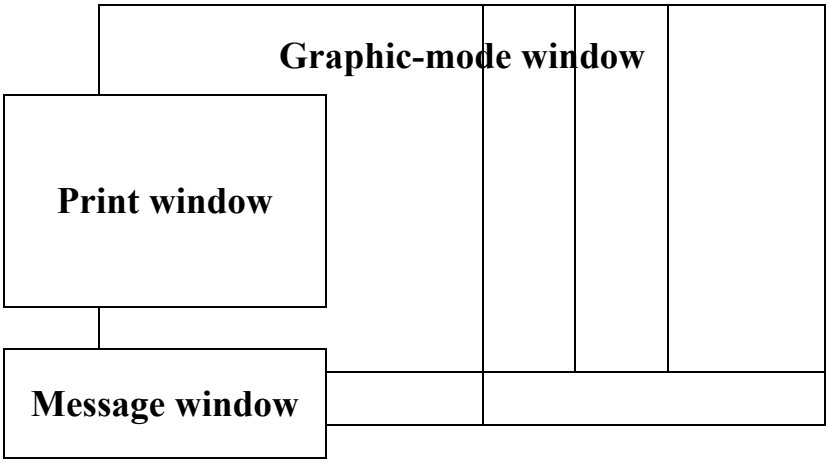


Figure 3.1 The Prefem windows on PC

On Unix there is only a line-mode window in addition to the graphic-mode window. I.e. the print and message windows are replaced by a line-mode window where print requested by the user as well as program messages appear. (The line-mode window is also where line-mode input is entered if you do not use the graphical user interface; see Section 4.1.4 on this.)

Prefem offers two modes of input and both are available in the graphic-mode window:

- Line-mode input, i.e. typing commands using the keyboard
- Graphic-mode input, i.e. selecting commands by clicking the left mouse button (LMB)

A sketch of the graphic-mode window is shown in Figure 3.2 together with explanations of the six different areas. The true appearance of the graphic-mode window is shown in Figure 3.6. How to use the areas is explained in more detail in the following.

You may at this stage decide to go through a brief tutorial. Go then to Section 3.2 and use the explanations of the areas of the graphic-mode window below for reference.

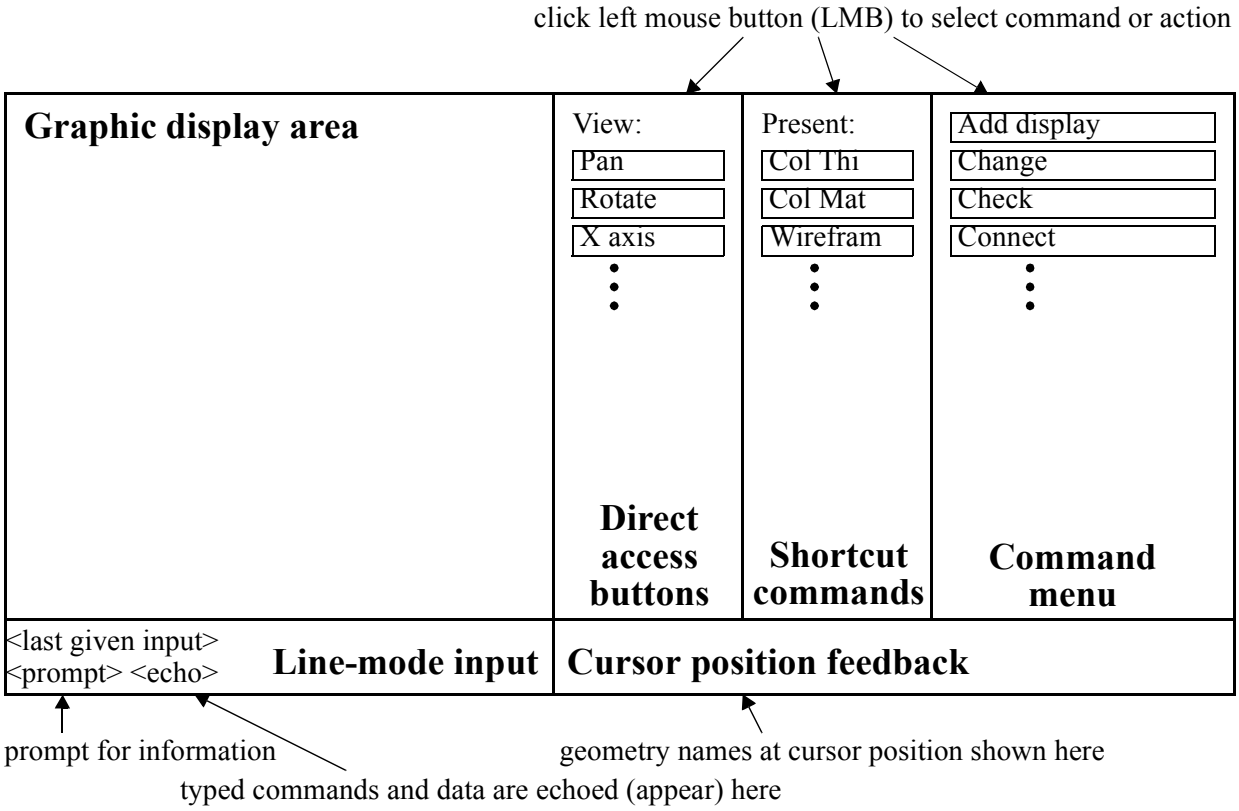


Figure 3.2 The graphic-mode window is composed of six different areas

The six different areas of the graphic-mode window are used as follows:

- **Graphic display area**
  - — The geometry and FE models are displayed here. The display is automatically updated when new geometry is defined. See Section 3.12.1 for examples of how to display the model.
  - — Within several commands there is a need for selecting geometry, e.g. when boundary conditions are defined and when loads are defined. Alternatively to giving line-mode commands as explained in Section 5.1 you may select geometry graphically. There are three ways of doing this:
    - Clicking the left mouse button (LMB)
    - Dragging a rubber-band rectangle using the LMB
    - Polygon selection: Position the cursor and press the shift key to define the first polygon point. While keeping the shift key pressed, repeatedly move the cursor and click the LMB to make a polygon. Release the shift key and click to define the last polygon point. A straight line between the first and last polygon points closes the polygon. If the LMB is pressed rather than clicked, a rubberband line appears as an aid to determine the position of the polygon segment.
  - — Apply the above graphical selection methods as follows:
    - Select a point and line/curve by the item itself or its label.

- Select a surface by rubberband/polygon around it or by clicking its label. You may also use the right mouse button (RMB) and left mouse button (LMB) in combination as follows: Click the RMB once on a borderline/curve of the surface and see that the line/curve is highlighted (changing colour). Without moving the mouse click the RMB once more and an adjoining surface is highlighted. If this is the desired surface then click the LMB. If not, then keep clicking the RMB until the desired surface is highlighted whereupon the LMB is clicked. Note that all clicks with the RMB and LMB should be done *without moving the mouse* (or at least not more than a set fractional distance).
- Select a body in the same way as explained above for a surface. When the RMB is repeatedly clicked, and after highlighting all adjoining surfaces, the adjoining bodies will be highlighted one after another.
- Note that you cannot 'loop' through the geometrical entities more than once, i.e. after highlighting the last body clicking the RMB will no longer have any effect.

— The availability of graphical selection is subject to that geometry selection has been switched on by the Direct access buttons Point, Line, Surface and Body. By default they are all switched on (depressed). See information on these buttons below.

— Note that if the Direct access button Info is depressed then geometry *cannot* be selected by clicking. See information on the Info button below.

- **Command menu**

— The at any time allowable commands plus default values for numerical data are listed here as buttons.

— Commands and values are selected by clicking the left mouse button (LMB).

— Slanted text signifies default choices that are accepted by either:

- Hitting the Return key
- Clicking either of the Direct access buttons ';' (semicolon) and '/ /' (double slash). The former accepts all subsequent default values (see Section 4.1.4) while the latter accepts a single default value, i.e. the one shown in slanted font.

- **Shortcut commands**

These provide one-click access to commonly used compound commands. A Shortcut command is logged as its equivalent full standard commands. The Shortcut commands are sorted in four groups as follows:

— Present:

- **Col Thi** — Colour shell/membrane elements according to their thicknesses
- **Col Mat** — Colour elements according to their material type
- **Wirefram** — Switch off colouring of elements, i.e. revert the actions of 'Col Thi' and 'Col Mat'
- **Hidden** — Toggle (switch on and off) hidden display (removing hidden lines)
- **Shrunken** — Toggle shrunken display (with factor 0.7)

— Display:

- **Geometry** — Display all geometry
- **Surface** — Display selected surfaces, the surfaces then need to be selected
- **Element** — Display selected elements, the elements then need to be selected
- **Mesh** — Display the FE mesh

- **Add Mesh** — Add display of the FE mesh to the currently displayed geometry
- **Add Load** — Add display of loads to the currently displayed FE mesh

— Label:

- **Boundary** — Toggle (switch on and off) symbols for boundary conditions
- **Points** — Toggle name of points
- **Lines** — Toggle name of lines
- **Surfaces** — Toggle name of surfaces
- **Geometry** — Toggle name of all geometry names (points, lines, surfaces and bodies)
- **Nodes** — Toggle node numbers
- **Elements** — Toggle element numbers
- **Supernod** — Toggle symbol (blue octagon) for supernodes
- **Mesh** — Toggle node and element numbers plus symbol for supernodes
- **All Off** — Switch off all labels (names, numbers and symbols)

— Default:

- **Mesh Adj** — Toggle (switch on and off) the feature: Whenever the number of elements (or maximum element size) is changed for lines do the following: Adjust mesh refinement starting with the lines next to the lines for which the discretisation has been changed and propagate outwards until a mesh for the whole model can be made. See the SET DEFAULT ADJUST-MESH and MESH ADJUST commands.

- **Direct access buttons**

  These buttons are accessible at any time. I.e. when you are in the middle of a command (by clicking a command or a Shortcut command or by typing a line-mode command) you may rotate and zoom to get a better view. The buttons ';' and '/ /' are logged with the default values they accept. The button '..' is logged as is. The other buttons are not logged (see Section 4.1.5 on logging commands). The Direct access buttons are sorted in three groups as follows:

  — View:

  - **Pan** — allows panning (shifting) the display. Click the button, then press and hold the LMB within the Graphic display area and a bounding box of the model appears. Move the mouse and release the LMB and the model will be displayed in its new position. (Note that the coordinate axes drawn together with the bounding box do not show the position of the origin of the model.)
  - **Rotate** — allows interactive rotation of the display. Click the button, then press and hold the LMB within the Graphic display area and a bounding box of the model appears. Move the mouse up and down to rotate the model about a screen horizontal axis and move left and right to rotate about a screen vertical axis. A circular motion will rotate the model about an axis normal to the screen in the *opposite* direction of the circular motion. When the LMB is released the model is displayed in its new position. (Note that the coordinate axes drawn together with the bounding box do not show the position of the origin of the model.)
  - **X axis**, **Y axis** and **Z axis** — display the model as seen along the model's X-, Y- and Z-axis, respectively.
  - **Default** — switches back to the default viewing position (optionally set in Manager) and re-displays the model.

- **Zoom In** — zooms in by either clicking twice and diagonally or by pressing the LMB and dragging it to form a zoom area (rubber-band box).
- **Zoom Fr** — re-displays the model so that it fits within the display area.
- **Refresh** — refreshes the display with the last setting.

— Misc:

- **Learn** — offers making a new Shortcut command. Click the button and enter a maximum eight character string being the name of the new Shortcut command and hit Return. Now give any sequence of commands. Several complete commands may be given, the last of which may be incomplete (i.e. more data is required to make it complete). Clicking the learn button once more completes the process and the new Shortcut command appears as a new button.
- **Info** — toggles (switches on and off) a mode offering quick information on geometric items. When the button is depressed clicking geometry provides information on the clicked items. This is typically point coordinates, line and surface properties, etc. In addition, the distance between points as well as angle between lines with a common end point are printed when points/lines are shift-clicked. The information appears in the print window (line-mode window on Unix). *Note that when the button is depressed geometry cannot be selected by clicking.* Dragging rubber-band still works as selection though.
- **;** — accepts all available default commands and parameters.
- **..** — aborts the current command.
- / / — accepts a single default value, i.e. the one shown in slanted font.

— Select:

- **Draw Geo** — displays the geometry. Note that as opposed to the DISPLAY GEOMETRY command (and the corresponding Shortcut command) this button is available within any command. This makes the button very convenient: e.g. if the mesh is displayed and you enter the PROPERTY LOAD command to define a load whereupon you realise that you need to have the geometry displayed to select where the load should be applied, then click the Draw Geo button and proceed.
- **Point** — must be depressed (the default condition) to allow graphical selection of points. Also, the Cursor position feedback, see below, only works for points when the Point button is depressed.
- **Line** — must be depressed (the default condition) to allow graphical selection of lines/curves. Also, the Cursor position feedback, see below, only works for lines when the Line button is depressed.
- **Surface** — must be depressed (the default condition) to allow graphical selection of surfaces. Also, the Cursor position feedback, see below, only works for surfaces when the Surface button is depressed.
- **Body** — must be depressed (the default condition) to allow graphical selection of bodies. Also, the Cursor position feedback, see below, only works for bodies when the Body button is depressed.
- **Set** — is merely a consequence of GUI consistency with other SESAM preprocessors and has little relevance for Prefem.

- **Line-mode input**

  — The upper line presents the last given input.

  — The lower line includes the prompt for input and data entered in line-mode.

  — On PC you may paste (Ctrl+V) text into the line-mode input area.

- **Cursor position feedback**
  - — The names of geometry at or close to the cursor position are listed here. If more than one geometric item is within the tolerance of the cursor position then all these will be listed in the sequence points, lines, surfaces, bodies.
  - — This Cursor position feedback only works when the Point/Line/Surface/Body Direct access buttons are depressed. This may be utilised as follows: If you cannot tell which is which of line and surface names because there are several names listed you may click (lift) the line (or surface) button. Then only surface names (or line names) will be listed.

Note: While entering a command by the keyboard it is not possible to click buttons or commands until hitting the Return key or deleting all data typed. This involves that if you (inadvertently) have entered a space character (which you may overlook as you cannot see it) clicking commands as well as selecting nodes and elements by clicking will not work. Hit the backspace a few times to delete the space character(s).

Note: Graphical selection of geometry does not work if the Info button is depressed. You will then instead get information on the geometry. See the explanation of the Info button above.

3.2 Tutorial in Midship Section Modelling

A tutorial based on using Prefem in graphic-mode is presented here.

Start Manager and open a new project. Then give Model | General Prefem (or click the appropriate tool button) and the General Modelling (Prefem start-up) window appears; see Figure 3.3. This window offers the following choices:

- The Database status will come up as New when Prefem is started the first time for a project. If you leave Prefem (the Prefem database will automatically be saved) and re-enter to continue modelling the Database status will come up as Old as Manager will detect the existing database. Changing Old to New in such a case involves deleting the existing database and creating a new and empty one, i.e. starting afresh.
- The Input mode box should always be Graphic.
- Section 4.1.2 explains the use of the Command input file box. In this case let it be None.
- By default the Write superelement on exit box will be checked. This involves that the Input Interface File containing the model (the file transferred to the analysis program) will be written when leaving Prefem by the EXIT command. If you foresee that you will not complete modelling in the current session you may uncheck this box. (But in the final session you need to check it.)
- Section 2.13.1 explains the Optimise superelement box. In this case let it be unchecked.

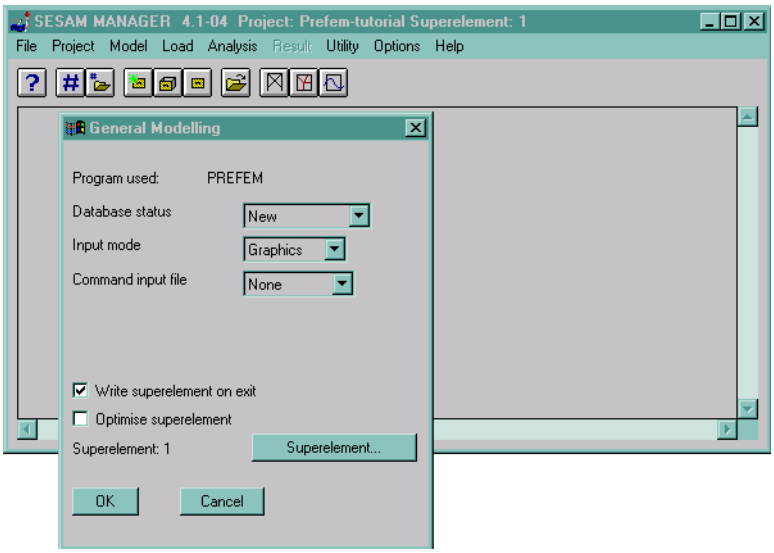


Figure 3.3 Manager and the General Modelling (Prefem start-up) window

Click OK in the General Modelling window and Prefem will start and the graphic-mode window will appear; see Figure 3.2 and Figure 3.6.

In this tutorial we will create a model of a midship section with geometry, dimensions and boundary conditions as shown in Figure 3.4. The desired FE mesh, plate thicknesses and loads are as shown in Figure 3.5. Units are metre, Newton and kilogram. Refer to Appendix A 1 for the full line-mode input.

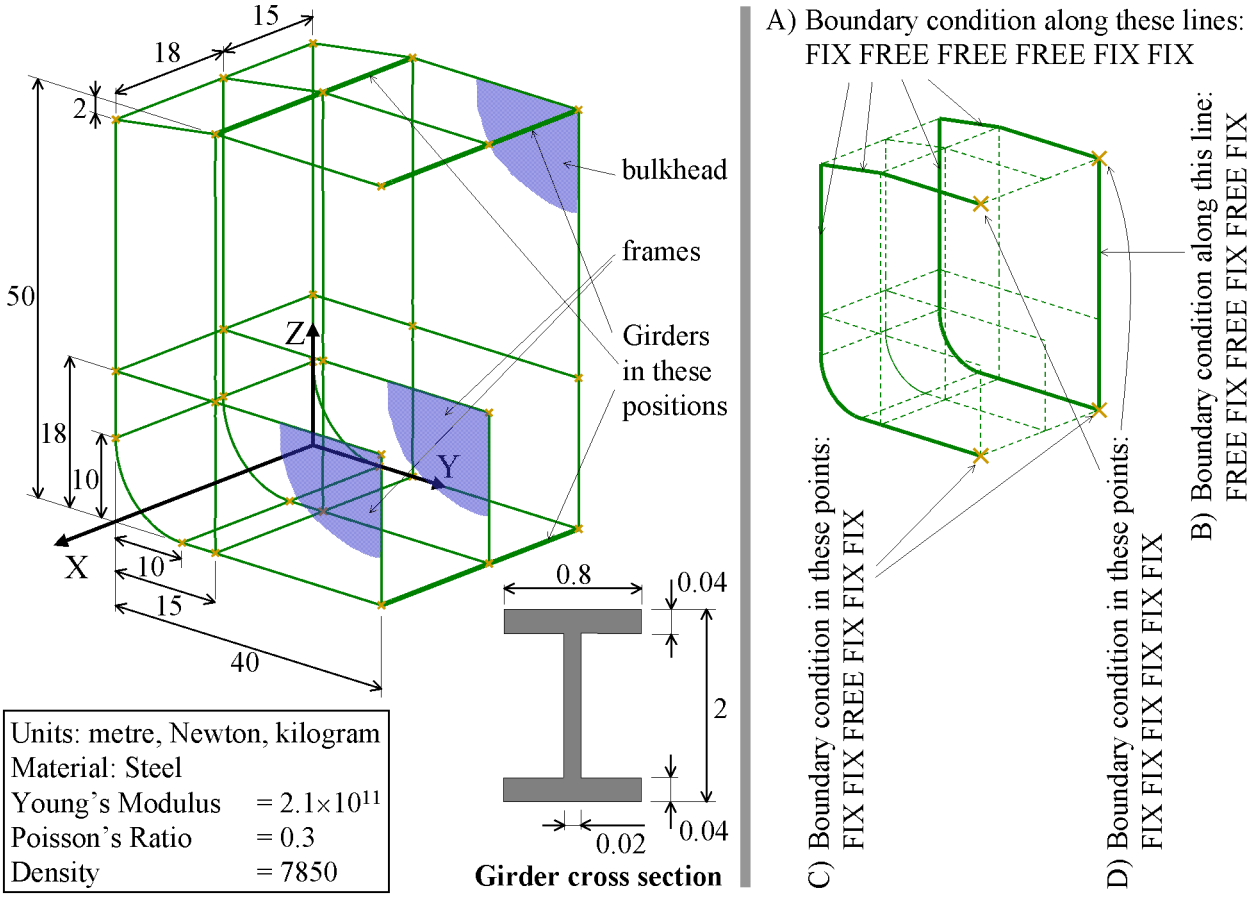


Figure 3.4 Tutorial — geometry with dimensions and boundary conditions

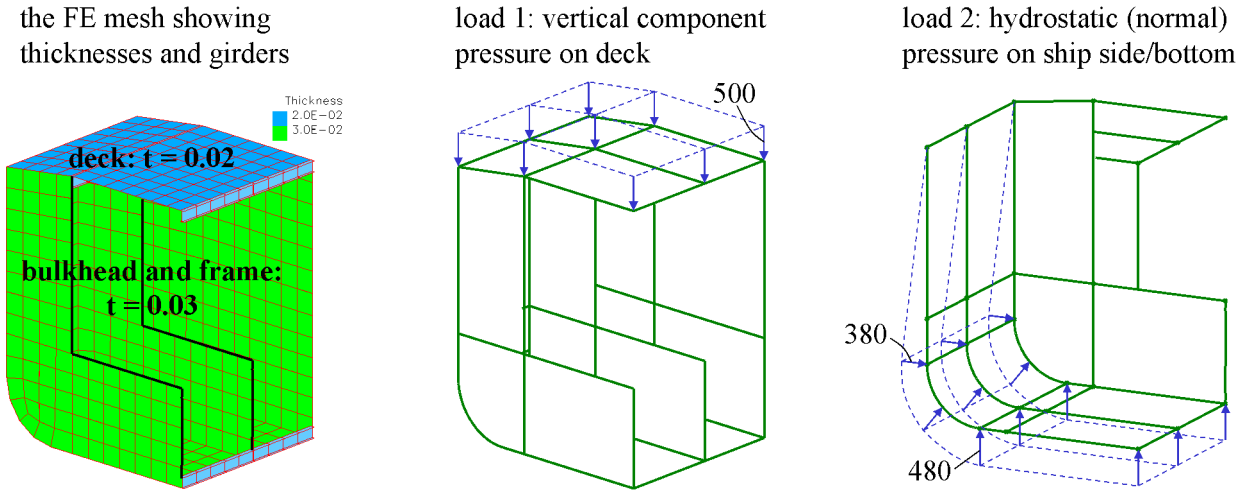


Figure 3.5 Tutorial — FE mesh, plate thicknesses and two load cases

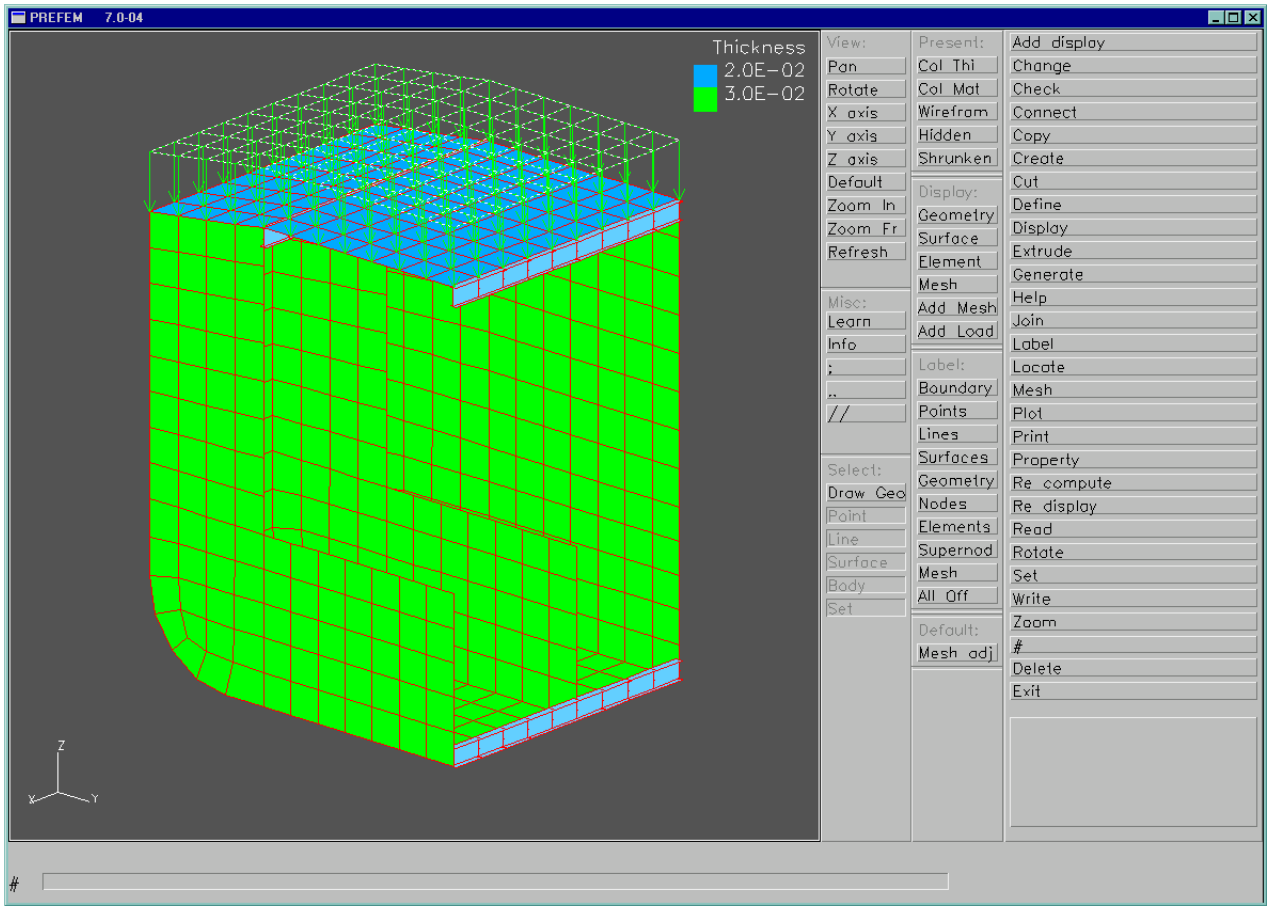


Figure 3.6 The graphic-mode window with the midship section model and load case 1

In the following the modelling procedure is described step by step while referring to line-mode commands. Some times the required command is give in full but generally only the first part is given assuming that the remaining part is self-explanatory. You may also refer to Chapter 5 where all commands are described in detail.

Rather than entering the line-mode commands through the keyboard you are in this tutorial advised to click the commands in the Command menu (the right most column). That way is probably quicker and you will at the same time learn about alternatives to the commands you are entering by examining the Command menu.

The modelling procedure step by step:

- Use the command SET DEFAULT AUTOMATIC-NAMING ON to let the program give names to geometry you define. Not having to specify geometry names the modelling will be quicker. Note that for more complicated models you may want to control the naming of geometry in order to take advantage of Prefem features based on a systematic naming (see Section 3.9.3).

- Use DEFINE POINT to define the four points shown in Figure 3.7. Having defined the points leave the DEFINE POINT command by typing '..' or by clicking the Direct access button '..'.

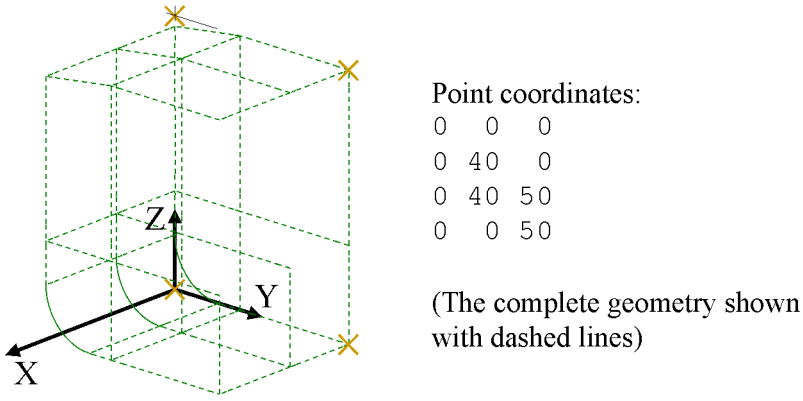
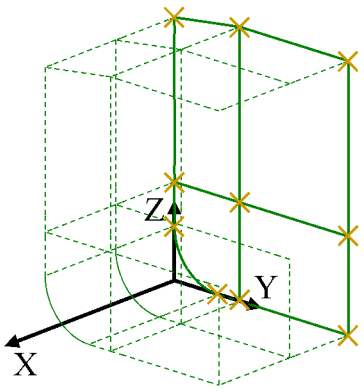


Figure 3.7 Tutorial — the first four points spanning the initial surface

- View the model along the X-axis by clicking the Direct access button ‘View: X axis’ (in the leftmost column).
- Give the command DEFINE SURFACE and click the four points to define a surface. Start at any of the four points, proceed in a rotational sequence and close the surface by a fifth click at the starting point.
- Generally, leave commands by typing ‘..’ or by clicking the Direct access button ‘..’. Complete datasets will be saved. Note, however, that certain commands are not complete until one or two END commands have been entered. The PROPERTY LOAD command is an example of such a command.
- Use CUT ALL-SURFACES-INCLUDED PREDEFINED-PLANE to cut first horizontally and thereafter vertically. The first cut is parallel with the XY-PLANE and at Z-coordinate 18 metres. The second cut is parallel with the XZ-PLANE and at Y-coordinate 15 metres.
- Refresh the display after the CUT commands by clicking either the Shortcut command ‘Display: Geometry’ or the Direct access button ‘Select: Draw Geo’. The difference between the two is that the latter may be clicked within any command, e.g. at the point where you want to select geometry by clicking. Note that after commands that change the current geometry (CUT, DEFINE ROUNDED-CORNER, DEFINE SECTOR-CORNER, CHANGE POINT) you should refresh the display before graphically selecting geometry. Refreshing the display is also relevant in order to remove highlighting of geometry caused by graphical selection.
- Use DEFINE ROUNDED-CORNER to round off the corner of the surface located in the origin (this will be the bilge). First click the two lines at both side of the corner, thereafter give the radius (10 metres) and conclude by requesting the corner (the area outside the arc) to be deleted in the rounding off operation.
- Use CHANGE POINT to move the upper left point (0,0,50) down 2 metres (this will form the sloping outer deck). After clicking the point use the ‘less than’ and ‘greater than’ bracket commands to subtract 2 from the Z-coordinate like this: < DZ -2 >.

At this stage the bulkhead should be complete and appear as shown in Figure 3.8 (apart from the viewing angle).



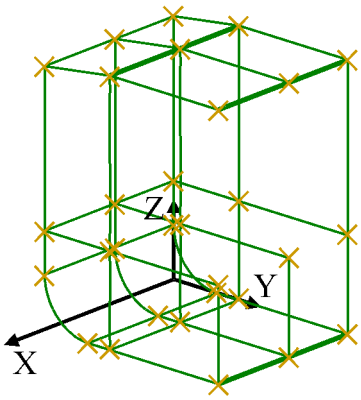
The bulkhead is completed

(The complete geometry shown with dashed lines)

Figure 3.8 Tutorial — the initial surface has been cut twice, corner rounded (bilge) and point moved

- The bulkhead should now be extruded to form the complete midship section. Give the EXTRUDE command proceed as follows:
  - — Select the three surfaces to copy to make the two frames (all surfaces but the upper right one). To select multiple surfaces start and conclude the selection by the left and right parentheses (these are commands on their own). Select a surface by clicking the right mouse button repeatedly on a borderline until the proper surface is highlighted whereupon the left mouse button is clicked (do not move the mouse during this operation). Such selection is also described under the explanation of the Graphic display area in Section 3.1. When concluding the selection by the right parenthesis command the three surfaces will automatically be displayed.
  - — Give prefix for names of copies of geometry, e.g. COP.
  - — Select the lines to extrude to make the deck and skin. To select multiple lines start and conclude the selection by the left and right parentheses. Select (by clicking) the two upper lines, the lines along the Y- and Z-axes and the arc (the bilge). Note that in order to be able to click the horizontal upper line (the deck) you must display the complete geometry by clicking the Direct access button ‘Select: Draw Geo’.
  - — Give prefix for names of extruded geometry, e.g. EXT.
  - — Give coordinate system in which the copying and extrusion shall be performed: the global cartesian system.
  - — Give number of copies/extrusions: 2.
  - — Give the two vectors defining the copying/extrusion: [15,0,0] and [18,0,0].
  - — Specify that lines are to extruded to surfaces. The copied/extruded geometry will automatically be displayed.

At this stage the geometry model should be complete and appear as shown in Figure 3.9. Rotate the model, for example by clicking the Direct access button ‘View: Default’ (a default set in Manager) or by giving the command SET GRAPHICS EYE-DIRECTION 1.1 1.0 0.5.



The geometry model is completed

Figure 3.9 Tutorial — the initial surface has been cut twice, corner rounded (bilge) and point moved

- Use the PROPERTY command to assign thicknesses to surfaces, i.e. plates. Since most surfaces (skin, bulkhead and frames) have thickness 0.03 you may assign this thickness to all surfaces and thereafter assign thickness 0.02 to the four surfaces constituting the deck. If you have a viewing angle as in Figure 3.9 selecting the deck surfaces may be done by dragging a rubberband.

- Use the PROPERTY MATERIAL command to define the steel material properties (see Figure 3.4 for data). Give a name (limited to 8 characters) to the material.

- Use the CONNECT MATERIAL command to assign the named steel material to all surfaces (the plates) and all lines (the girders).

- Use the SET ELEMENT TYPES command to assign element types to geometry (the subsequent MESH command creates the actual elements). First assign element type to surfaces: All surfaces shall be assigned the 4 node shell (SHELL-4NODES). Then assign element type to lines: Lines where there are girders (see Figure 3.4) shall be assigned 2 node beam (BEAM-2NODES).

- Use the PROPERTY SECTION command to define the girder cross section (see Figure 3.4 for data). Give a name (limited to 8 characters) to the section.

- Use the CONNECT SECTION command to assign the named cross section to all lines (the assignment will only take effect for lines for which a beam element type has been assigned).

- Use the PROPERTY ECCENTRICITY-BEAM command to introduce eccentricities (offsets) for lines where there are girders. By default the neutral axis of a beam element extends from node to node and coincides with the middle surface of a plate element. But girders are welded onto the plates. All beam elements therefore require an eccentricity of half the plate thickness plus half the girder height. Rather than giving these eccentricities manually use the CALCULATED-NEGATIVE-Z-OFFSET and CALCULATED-POSITIVE-Z-OFFSET options. Given that in this case the local z-axes of the girders are parallel with the global Z-axis (see the PROPERTY LOCAL-COORDINATE-BEAM for more information on this) the girders of the deck must be moved in the negative local z-direction (down) while the centreline girder must be moved in the positive local z-direction (up).

- Use the SET MAX-ELEMENT-LENGTH command to set the maximum element length for all surfaces (and implicitly all lines) to 4.

- Use the MESH ALL command to create the FE mesh, i.e. nodes and elements.
- Click the Shortcut command ‘Display: Mesh’ to display the mesh and see that there are triangular elements in the bilge area of the bulkhead and frames.
- To improve the mesh in these areas do as follows:
  - — Use the SET DEFAULT ADJUST-MESH ON command (or click the Shortcut command ‘Default: Mesh adj’) to set Prefem in a mode in which a change in mesh density (number of elements along lines) involves a re-meshing. If necessary Prefem will automatically adjust the mesh density other places too to be able to create a mesh for the whole model.
  - — Use the Shortcut command ‘Display: Geometry’ to display the geometry.
  - — Use the SET NUMBEROF-ELEMENTS command to reduce the number of elements along the three short lines parallel with the Y-axis and next to the bilge from 2 to 1. Remember to start and conclude the selection of the three lines with left and right parentheses.
- Click once more the Shortcut command ‘Display: Mesh’ to display the mesh and see that the meshes in the three areas have improved.
- Use the PROPERTY BOUNDARY-CONDITION command to define boundary conditions:
  - — First click the Direct access button ‘Draw Geo’ to display the geometry and allow you to select geometry.
  - — Thereafter select the lines/curves (remember parentheses) identified by an A in Figure 3.4 and give the boundary conditions FIX FREE FREE FREE FIX FIX for the six degrees of freedom (translations in X, Y and Z and rotations about the same).
  - — Then select the vertical line of the bulkhead midship identified by a B in Figure 3.4 and give the boundary conditions as specified.
  - — Select the two end-points of the centre-line girder (C in Figure 3.4) and give the appropriate boundary conditions.
  - — Finally, select the two points in the deck centre-line (D in Figure 3.4) and give the appropriate boundary conditions.

  Note that the boundary conditions given for the points will replace those previously given for the lines (the points are encompassed by the selected lines). It is therefore necessary to give boundary conditions for the points at the end.
- Use the PROPERTY LOAD command to define loads (see Figure 3.5):
  - — Load case number 1 is a COMPONENT-PRESSURE meaning that three components of the pressure shall be given: X, Y and Z. After selecting the four deck surfaces and choosing the global cartesian coordinate system for the load give zero for the X- and Y-components and -500 for the Z-component. Following the Z-value give END meaning that the load is not going to be complex (we are making a model for a static analysis and not for a frequency domain analysis). Conclude the load definition by specifying where on the plate the load shall apply: the MIDDLE-SURFACE (this choice has consequence for curved 6 and 8 node elements only). Give END to leave definition of load case 1.
  - — Load case number 2 is a NORMAL-PRESSURE meaning that only one component which is always normal to the plate elements is given. Select all skin surfaces. Being a hydrostatic pressure with the water surface at the level of the top of the ship side (Z=48) the load varies linearly in the vertical direction: zero pressure at Z=48, 380 at Z=10 (where the bilge starts) and 480 at Z=0 (ship bottom). This variation is easily defined by selecting the option LINEAR-2POINTS-VARYING and giving the

appropriate values for any two points on a vertical line. For example: Click the point in (0,0,48) and give the pressure value 0, then click the point in (0,0,10) and give the value 380 (linear extrapolation outside the two selected points). Then give END meaning that the load is not going to be complex. Conclude the load definition by specifying where on the plate the load shall apply: the MIDDLE-SURFACE. Give END to leave definition of load case 2.

- Display the mesh (Shortcut command 'Display: Mesh') and then click the Shortcut command 'Display: Add Load' and give load case 2. Hit Return once more and the load case will be displayed.

- Notice that for the two curved surfaces in the bilge area the pressure points in the wrong direction. This is caused by that the element normal points in the wrong direction for these surfaces. (With the mesh displayed you may use LABEL ELEMENT-NORMAL ON to see the element normals.) The direction of the element normals is a function of the way the surfaces were defined. To change the direction of the element normals and therefore also the pressure do as follows:
  — Use the DELETE MESH ALL command to delete the current mesh.
  — Click the Shortcut command Geometry (or the Direct access button Draw Geo).
  — Use the CHANGE ROTATION-OF-SURFACE command and select the two surfaces in question.
  — Use the MESH ALL command to re-create the mesh.

- Display the mesh once more and add display of load case 2 and see that the pressure now points in the proper direction.

The model should now be complete. Use the SET GRAPHICS PRESENTATION BEAM-ELEMENT SECTION-AS-SOLID command to display the girders with their sections shown and in their eccentric position. Try out the various Shortcut commands (for example 'Present: Col Thi' and 'Present: Col Mat') and Direct Access buttons while referring to the descriptions of these in Section 3.1.

If you want to perform a static analysis of the model and view the results do as follows:

- Leave Prefem by clicking EXIT. Provided that the box 'Write superelement on exit' in the General Modelling window (by which Prefem is started) was checked when entering Prefem the Input Interface File (see Section 2.13.1) will now be written. If this box was not checked then check it, enter Prefem once more and exit.

- Run Sestra from Manager, remember to choose the Multifront equation solver.

- Start Xtract from Manager, selecting Default Command input file will present the displacement of result case 1.

You may now want to learn more about geometry modelling, FE model creation, defining and assigning properties as well as defining loads. Section 3.3 through Section 3.6 deal with these topics.

3.3 Geometry Modelling

A model will always have a cartesian coordinate system to which the geometry will refer. A geometry model consists of the entities points, lines/curves, surfaces and bodies.

Geometry is defined by:

- Direct definition of geometrical entities (using the command DEFINE)
- Copying previously defined geometry (using the command COPY)
- Generation of 'regular' geometry (using the command GENERATE)
- Extruding and copying geometry (using the command EXTRUDE)
- Cutting (command CUT) and joining (command JOIN) existing geometry thereby modifying it
- Rounding off corners (command DEFINE ROUNDED-CORNER) and cutting a sector or hole in a corner (command DEFINE SECTOR-CORNER) thereby modifying existing geometry
- Importing geometry from a DXF file, a drawing exchange format supported by many CAD systems

All geometric entities have unique names. The names have a maximum length of 8 characters and start with any letter (names starting with X are reserved for program generated names). The names are given by the user when defining the geometry or, optionally, automatically assigned by the program. All reference to existing geometry is made by either referring to these names or by selecting geometry by graphical means.

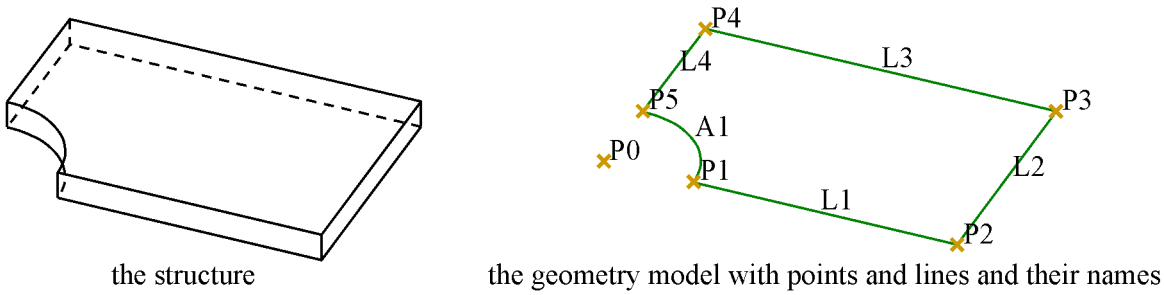


Figure 3.10 The geometry model of a structure

3.3.1 Defining Geometry

Defining Points

A point is the simplest geometrical entity. It is described by coordinates in the cartesian coordinate system of the model and defined by the DEFINE POINT command. The coordinates are either given directly, given relative to other points, calculated as an interpolation between two points, or calculated as an intersection between three shapes. Even though a model always will have a cartesian coordinate system cylindrical and spherical coordinate systems may be used for defining the points. A few examples of how to define points are given below.

The two commands below first define point P1 and then P2 relative to P1 by adding 10 in X-direction and subtracting 2 in Y-direction. Figure 3.11 shows the two points defined.

```
DEFINE POINT P1 2 6 4
DEFINE POINT P2 < P1 DX 10 DY -2 >
```

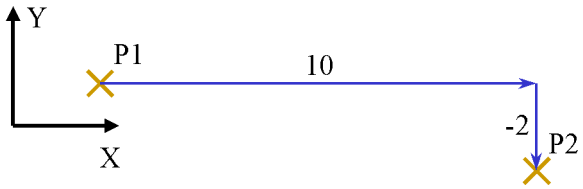


Figure 3.11 Defining points

The command parameter '<' opens an *update mode* for specification of point coordinates. Having given the command:

```
DEFINE POINT P2 <
```

the *current coordinates* (the last given) are echoed on the screen, e.g.:

```
X= 2.000E+00, Y= 6.000E+00, Z= 4.000E+00
```

The following options are then available for updating (changing) these initial coordinates:

- another-point-name
- X
- Y
- Z
- DX
- DY
- DZ
- SHAPE-INTERSECTION
- POINT-INTERPOLATION
- MOVE-BY-TRANSFORMATION
- USE-LOCAL-COORDINATE-SYSTEM

The X, Y, Z options followed by a value will replace the corresponding coordinates. The DX, DY, DZ options will add a given value to the corresponding coordinates. After each update the current coordinates are echoed on the screen. When satisfied the command parameter '>' is given to confirm the point coordinates and close the update mode.

Within this update mode real values may also be entered as expressions, see Section 3.5.10, and parameters, see Section 3.8.

Of the other updating alternatives listed above the POINT-INTERPOLATION and SHAPE-INTERSECTION are exemplified below and illustrated in Figure 3.12.

P3 is defined by interpolating between P1 and P2:

```
DEFINE POINT P3 < POINT-INTERPOLATION P1 P2 0.3 >
```

P4 is defined by intersecting three pre-defined shapes (two planes and a sphere). Note that the coordinates to update by the shape intersection, the initial coordinates of P4, must be in the vicinity of the intersection sought in order to pick the proper intersection point (if there is more than one).

```
DEFINE POINT P4 < SHAPE-INTERSECTION SH1 SH2 SH3 >
```

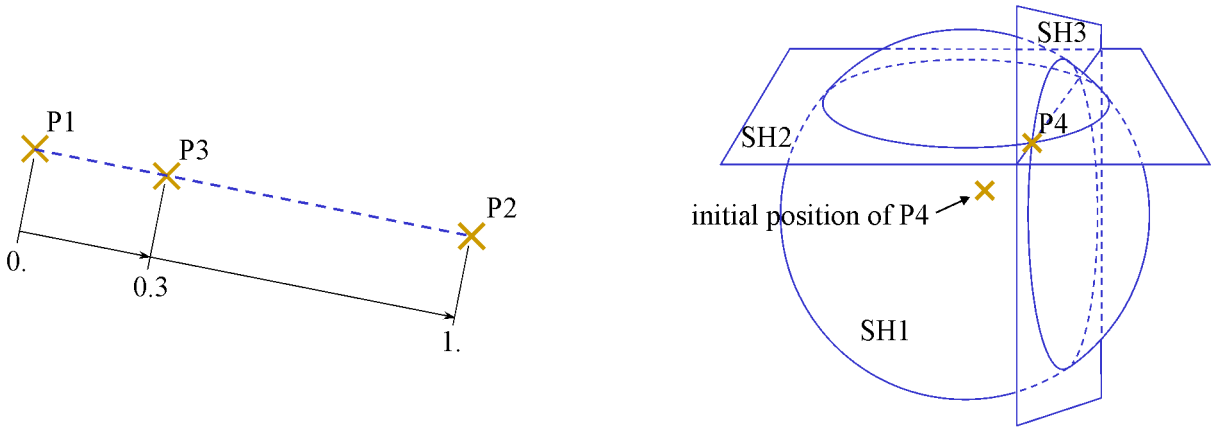


Figure 3.12 More on defining points

Defining Lines/Curves

A line/curve is defined by a start and an end point plus a rule to determine the shape of the line/curve. Table 3.1 shows these definitions for the various types of line/curve. How to define lines and curves is exemplified below and illustrated in Figure 3.13 through Figure 3.15.

Note: Mathematically, all lines and curves are described as B-splines between start and end points. A B-spline is a combination of polynomial functions and is capable of representing any complex curve. A circular arc is therefore not exactly an arc but rather a B-spline very close to the arc.

Table 3.1 Lines/curves and their definitions

Type	Definition
Line	Straight line between two points
Arc	Circular arc defined by two points and a centre point
Intersection	Intersection between two shapes and between two points, a third point picks the proper segment
Spline	B-spline curve defined by an arbitrary number of points

Table 3.1 Lines/curves and their definitions

Type	Definition
Node-line	Broken straight lines between an arbitrary number of points and with nodes in the points
Curve	Curve generated by GENERATE command in cylindrical or spherical coordinate system

The command below defines the straight line L1 between points P1 and P2. It will be discretisised into 4 elements or element-edges during meshing.

```
DEFINE LINE L1 P1 P2 4
```

The command below defines the circular arc A1 between P1 and P2 and with P0 as centre point. A1 will be discretisised into 5 elements.

```
DEFINE ARC A1 P1 P2 P0 5
```

The command below defines the intersection curve C1 between the two shapes SH1 and SH2. The endpoints of the curve are PA and PB. PX is a point in the vicinity of the desired one of possibly more intersection segments. C1 will be discretisised into 3 elements.

```
DEFINE INTERSECTION C1 PA PB PX SH1 SH2 3
```

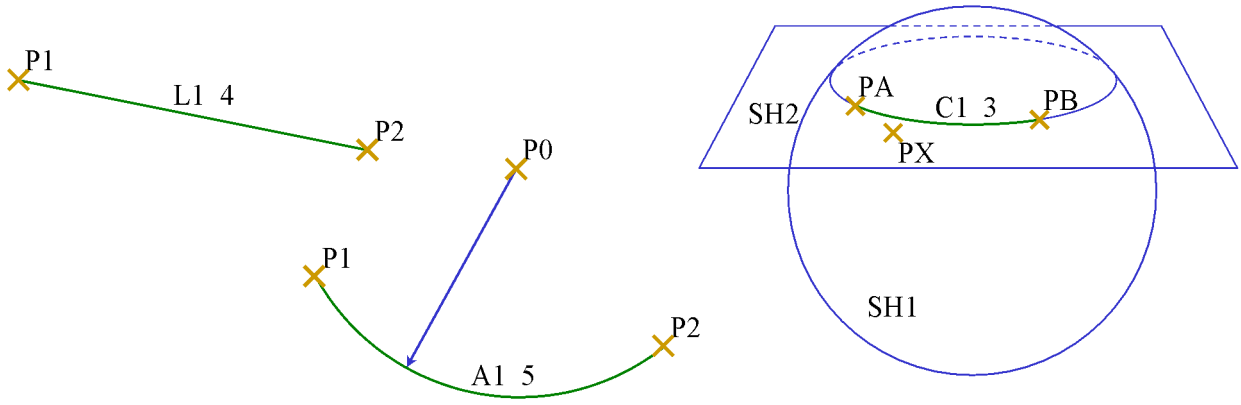


Figure 3.13 Defining line, arc and intersection

The command below defines the B-spline curve C2 going through the points P1 through P4. Note that a B-spline curve is often best defined if its ends are determined by two closely located points (P1-P1X and P4-P4X). C2 will be discretisised into 8 elements.

```
DEFINE SPLINE C2 P1 P1X P2 P3 P4X P4 8
```

The command below defines the node-line L2 going through the points P1 through P4. L2 will be discretisised into as many elements as there are line segments, in this case 3, and there will be a node in each point.

```
DEFINE NODE-LINE L2 P1 P2 P3 P4
```

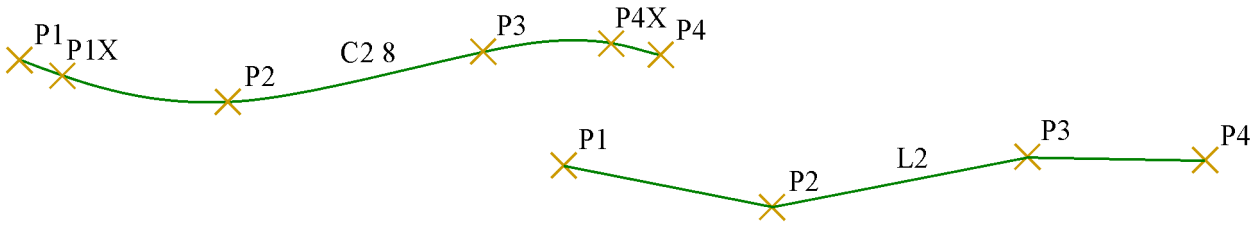


Figure 3.14 Defining spline and node-line

An additional feature related to defining lines/curves should be noted: A line/curve defined as one type, e.g. a line, may be changed into another type, e.g. an arc. The principle for changing type is to use the CHANGE command as if the line/curve is already of the desired type. The CHANGE command below changes a line into an arc by 'pretending' that the line is already an arc; Figure 3.15 illustrates the result. This feature is convenient when the GENERATE command (see Section 3.3.7) has been used to create a geometry consisting of straight lines (GENERATE in a cartesian coordinate system) while in fact some of the lines are arcs or splines. See Chapter 5 on CHANGE ARC / INTERSECTION / LINE for more information.

```
DEFINE LINE L1 P1 P2 4
CHANGE ARC L1 P1 P2 P0 4
```

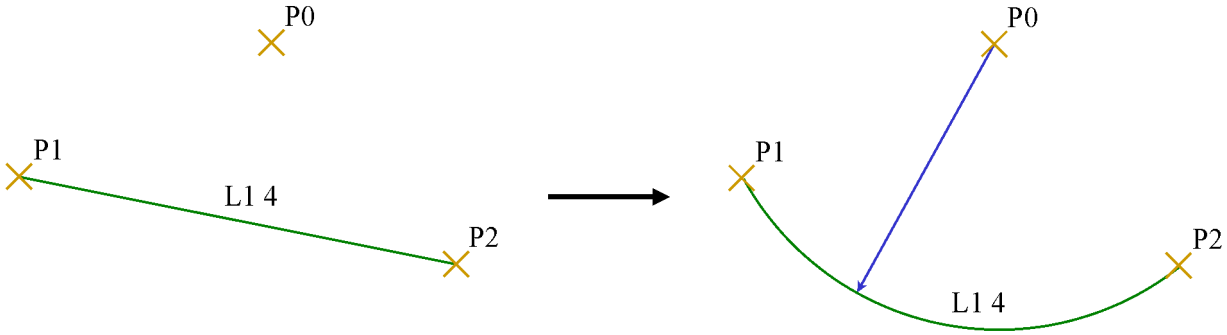


Figure 3.15 Changing line to arc

Defining Surfaces

A surface is defined by an unlimited number of borderlines and curves forming a closed chain. How to define various surfaces is exemplified below and illustrated in Figure 3.16 and Figure 3.17.

Note: Unless all borderlines and curves lie in a plane the shape of the interior of a surface will not be explicitly described. A *shape* may then be used to define the interior by projecting the surface onto the shape, for example a sphere or a cylinder. Thus, the border of a surface is defined by its borderlines and curves while its interior is defined by a shape. See Section 3.3.2 on shapes and Section 3.4.5 on element meshes for curved surfaces.

A surface may be defined by selecting any number of points on its border and concluding the list of points either by END or by closing the surface by repeating the first point selected. Straight borderlines with names

LI0, LI1, LI2, etc. will automatically be created where there are no lines. The command below defines surface S1. Note that a straight line will be created even in cases where a curve (e.g. an arc) instead of a line has already been defined; this may not be what the user wants. To avoid this pitfall and also to have full control over the line names define the lines/curves explicitly and thereafter define the surface by referring to the lines/curves instead of the points; see the next example (defining surface S2).

```
DEFINE SURFACE S1 P1 P2 P3 END
```

The above is equivalent to:

```
DEFINE SURFACE S1 P1 P2 P3 P1
```

The command below defines surface S2 bounded by the lines L1, L2 and L3. The program will automatically detect that the three lines enclose a surface (L3 joins L1) and will therefore not request or expect more lines.

```
DEFINE SURFACE S2 L1 L2 L3
```

The command below defines surface S3 bounded by the two *unconnected* lines L1 and L2. Unless already existing two straight lines with default names will be created between the end points of L1 and L2.

```
DEFINE SURFACE S3 L1 L2
```

The command below defines surface S4 bounded by the lines/curves L1, A2, L3, L4 and L5. By default, all five points will be *mesh-corners* (see Section 3.4.1 for a description of the concept of mesh-corners). Reduce the number of mesh-corners to the maximum allowable 4 by defining at least one of the points as a *not-mesh-corner*. Figure 3.31 illustrates a mesh that may have been created for surface S4.

```
DEFINE SURFACE S4 L1 A2 L3 L4 L5 ;
SET MESH CORNER-TYPE S4 P5 NOT-CORNER END
```

The two commands above may alternatively be substituted by the single command below.

```
DEFINE SURFACE S4 L1 A2 L3 L4 NOT-MESH-CORNER L5
```

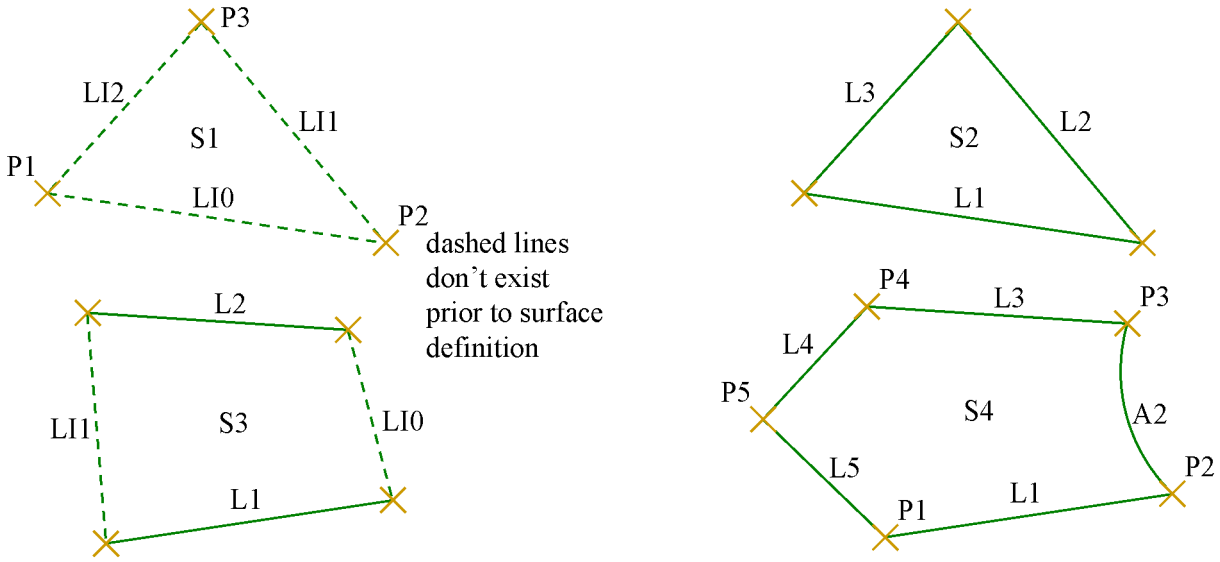


Figure 3.16 Defining surfaces

The command below defines a surface bounded by three arcs. As the arcs do not lie in a common plane the interior of the surface is not explicitly defined. In this case the mesh iteration process will give a concave mesh as shown in the middle of Figure 3.17. See Section 3.4.5 for more information on meshes for curved surfaces.

```
DEFINE SURFACE S5 A1 A2 A3
```

To get the spherical mesh as shown to the right in Figure 3.17 the interior of the surface must be defined by projecting the surface onto a spherical shape. This is done either within or after the command defining the surface. These two alternatives are exemplified below – SH1 is the name of a spherical shape with its centre and radius equal to those of the arcs.

Alt. 1:

```
DEFINE SHAPE SPHERE SH1 P0 radius ;
DEFINE SURFACE S5 A1 A2 A3 ;
SET PROJECTION S5 SH1
```

Alt. 2:

```
DEFINE SHAPE SPHERE SH1 P0 radius ;
DEFINE SURFACE S5 SH1 A1 A2 A3
```

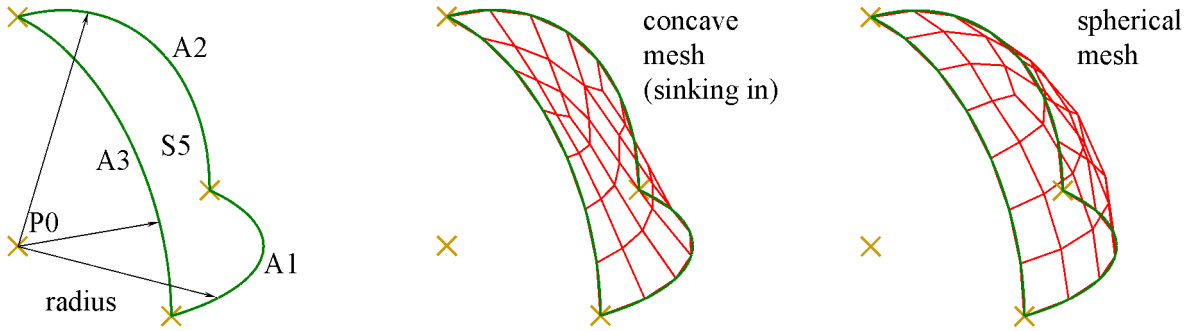


Figure 3.17 Defining surfaces by projecting onto a shape

Defining bodies

Solid objects are modelled as bodies which by definition are enclosed by top and bottom surfaces and any number of side surfaces. The following requirements to a body and its mesh must be met:

- A body must logically, if not geometrically, be prismatic as shown in Figure 3.18.
- This implies that the side surfaces must be quadrilateral, if not rectangular, and extend from the bottom surface to the top surface. In the command defining bodies the side surfaces must be given in sequence.
- The top and bottom surfaces must have been defined in the same sequence, starting and ending in corresponding positions (points/lines). The number of borderlines of the top and bottom surfaces need not be the same but the surfaces must have mesh-corners and not-mesh-corners in corresponding positions. This means that if, say, two lines of the bottom surface correspond to one line of the top surface then there must be a not-mesh-corner in-between the two lines of the bottom surface. See Figure 3.18.

- The element mesh for a body must also be prismatic, i.e. the meshes of the top and bottom surfaces must in terms of topology be equal (their corresponding borderlines/curves must have equal number of elements and they must have mesh-corner and not-mesh-corner in corresponding points). Further, the mesh of the side surfaces must be regular with no mesh refinement and with mesh-corners in the four points connecting with the top and bottom surfaces and only there.

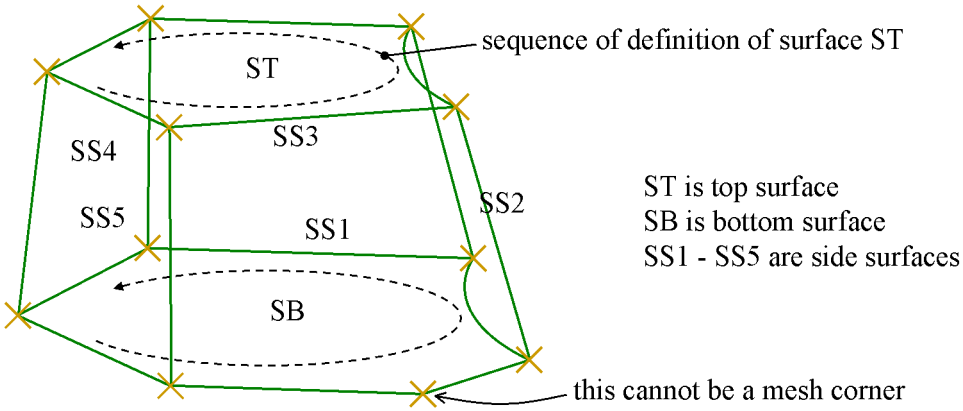


Figure 3.18 Requirements to a body

There are two commands for defining bodies:

- DEFINE BODY
- DEFINE PRISM

The difference between these two commands is that in the latter only the top and bottom surfaces are defined beforehand. The DEFINE PRISM command thus involves implicit and automatic definition of the side surfaces by generation of straight lines between corresponding points of the top and bottom surfaces.

3.3.2 Defining Shapes

Shapes are tools in the form of surfaces used for defining geometry and for projecting FE mesh onto them. In itself a shape does not constitute a part of neither the geometry model nor the FE model. The DEFINE command is used for defining shapes.

The different shapes are given in table Table 3.2 and illustrated in Figure 3.19.

Table 3.2 Shapes and their definitions

Shape	Definition
Plane	Three points
Sphere	One point and radius
Cylinder	Two points and radius
Cone	Two points and two radii

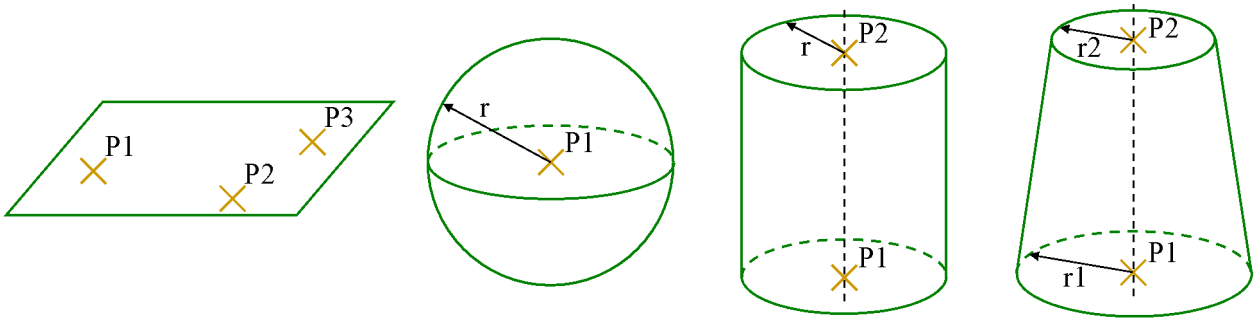


Figure 3.19 Shapes

Section 3.3.1 includes a few examples of use of shapes for defining geometry.

3.3.3 Copying Geometry

Copying geometry is performed by the COPY command and referring to transformations defined by the DEFINE TRANSFORMATION command. The following types of transformation may be defined:

- Translation
- Rotation
- Scaling
- Mirroring

A transformation may also be a combination of these. Be aware of that the order of the transformations generally is of consequence: A translation followed by a rotation is in general different from the same rotation followed by the same translation. See Section 3.11.1 for more information on transformations.

Any defined point, line, surface or body can be copied using the COPY command. The command specifies the geometry to be copied, the name of the copy and the transformation defining the copying process.

The command below copies the line LI1 plus the points belonging to the line by referring to the transformation T1. T1 must therefore first be defined. The string '&&2' in the COPY command specifies the names of the copies — the two points and the line — as follows: The two ampersands (&&) involve that the two first characters of a copy name will be taken from its origin while the third character will be replaced by 2. See Figure 3.20. See Section 3.9.3 for information on the concept of wild-cards.

```
DEFINE TRANSFORMATION T1 TRANSLATE DISPLACEMENTS 0 0 -10 ;
COPY LI1 &&2 T1
```

When copying geometry some properties are carried over from the original to the copy and some are not. See the description of the COPY command for more details on this.

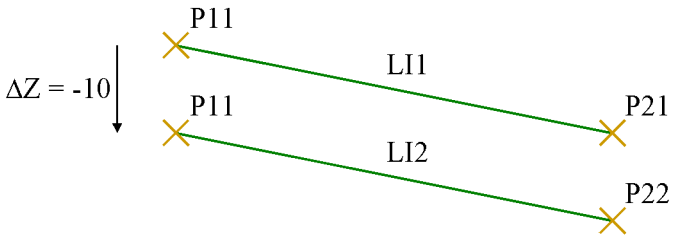


Figure 3.20 Example of use of the COPY command

3.3.4 Cutting Geometry

The geometry model may be cut by the CUT command. As illustrated in Figure 3.21 a surface S1 may in alternative ways be cut in two resulting in two new surfaces (SU0 and SU1), a new line along the split and each of the two bounding lines cut in two as well. The figure also illustrates how the command will attempt to distribute the number of elements set for lines according to where the lines are cut.

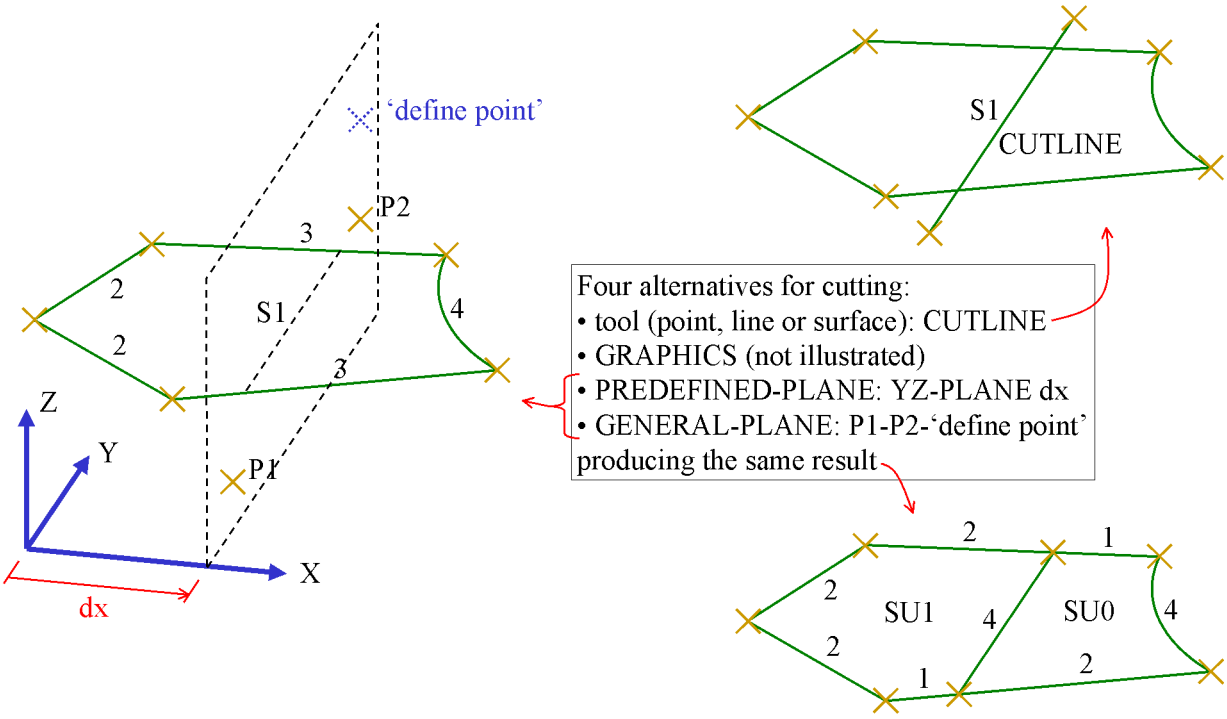


Figure 3.21 Cutting geometry

Plane surfaces will be cut by straight lines.

On certain conditions cylindrical and spherical surfaces may be cut:

- Surfaces created by the GENERATE command using a cylindrical or spherical coordinate system (which involves that the cylindrical/spherical coordinate system will be assigned to the surface).

- Surfaces to which a cylindrical/spherical coordinate system has been assigned by the SET MESH COORDINATE-SYSTEM command.
- Surfaces projected onto a cylindrical or spherical shape.

The new surfaces resulting from the cutting will inherit the assignment to coordinate system, alternatively the projection onto a shape, from the original surfaces.

Note: Surfaces projected onto shapes of type cone and interpolation will get straight cut lines.

Note: Any non-plane surface not satisfying the conditions above will get straight cut lines.

3.3.5 Joining Geometry

The JOIN command allows joining bodies.

Note: Surfaces cannot be joined. You cannot revert the cutting of a surface by joining the two halves.

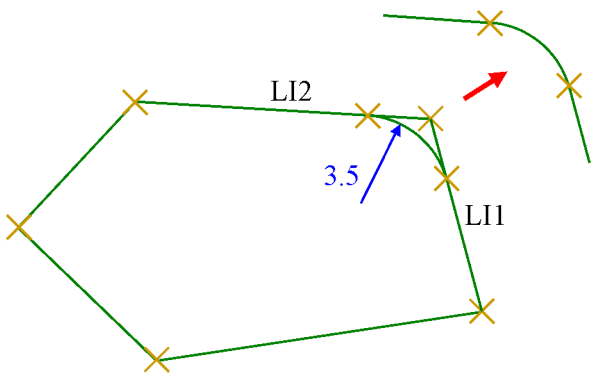
3.3.6 Rounding off Corners and Cutting Holes

The DEFINE ROUNDED-CORNER command rounds off corners as illustrated in Figure 3.22. Alternatively to deleting the corner, as shown in the figure, the corner complement (the whole surface but the corner) may be deleted and none of the two surfaces may be deleted.

The DEFINE SECTOR-CORNER command cuts holes in surfaces as illustrated in Figure 3.22. Alternatively to deleting the corner, as shown in the figure, the corner complement (the whole surface but the corner) may be deleted and none of the two surfaces may be deleted. The command will after selecting one surface and choosing which surface to delete allow repeating the process for the neighbouring surfaces thereby completing the process of making a round hole.

Round off a corner by selecting (clicking) lines on both sides and giving radius, then delete corner:

```
DEFINE ROUNDED-CORNER LI1 LI2 3.5
  CORNER
```



Make hole by giving radius, selecting (clicking) point, selecting surface and delete corner, selecting next surface and delete corner, etc.:

```
DEFINE SECTOR-CORNER 3.5 P1
  S1 CORNER
  S2 CORNER
  etc.
```

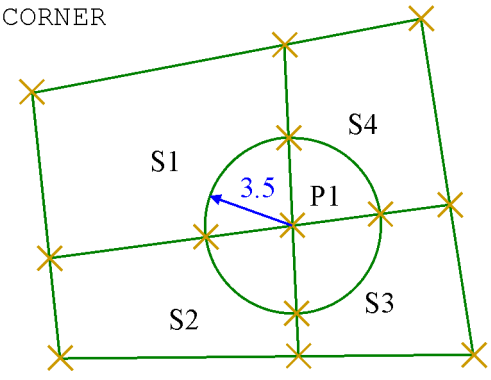


Figure 3.22 Rounding off corners and cutting holes

3.3.7 Generating Geometry

See Section 3.3.9 for a brief discussion on the GENERATE command versus the EXTRUDE command.

The GENERATE command is available for highly efficient generation of *regular* geometric shapes. A regular geometry is (in a cartesian system) a geometry consisting of surfaces all shaped as parallelograms. The command is based on first defining a topological I-J-K space and then, by use of vectors, mapping this space into a geometrical coordinate system. See Figure 3.23 for an illustration of this mapping technique for a 2-D space (only I-J). The requirement — or rather an inescapable consequence of the command — that all surfaces have the shape of parallelograms is also shown.

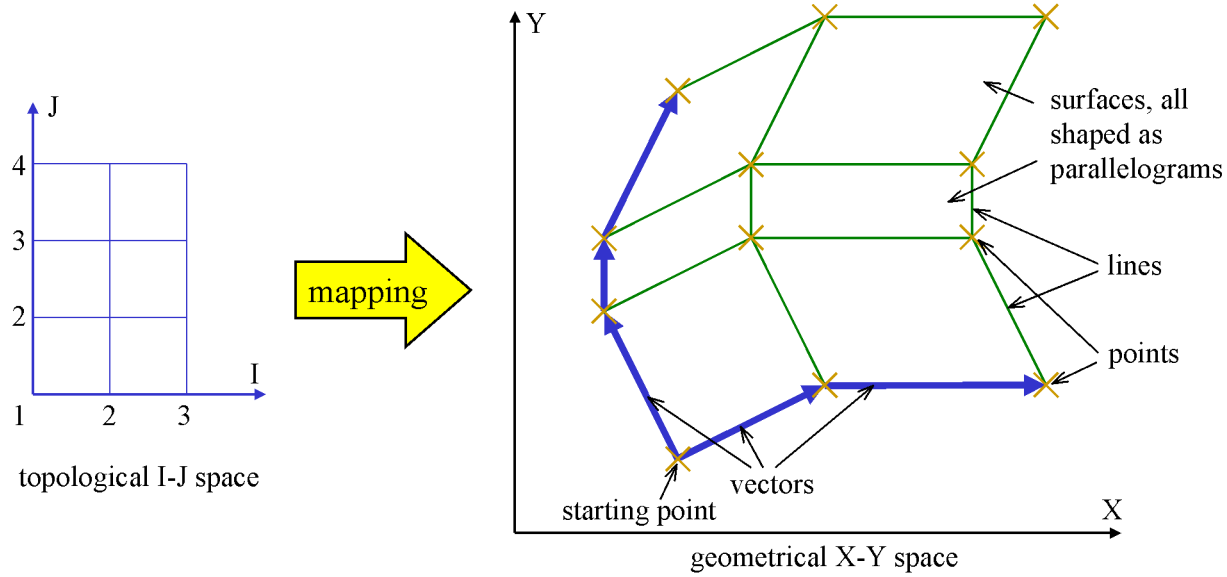


Figure 3.23 The GENERATE command maps a topology space into a geometry space

A single GENERATE command will typically replace a number of commands for defining points, lines, surfaces and (if relevant) bodies, i.e. a number of DEFINE POINT ..., DEFINE LINE ..., DEFINE SURFACE ... and DEFINE BODY ... commands. The structure of the command is as follows:

GENERATE level id topology coordinate-system geometry

Except for ‘level’ and ‘id’ the parameter names above each represent several parameters as detailed in the following. The GENERATE command is described in principle below; for a complete description see the command description in Chapter 5.

level Determines the ‘level’ of the geometry to be generated. The options are:

- POINT

- LINE

- SURFACE

- BODY

Generating lines involves generating points and lines, generating surfaces involves generating points, lines and surfaces, and, finally, generating bodies involves generating points, lines, surfaces and bodies.

id — An identification of the geometry to generate in the form of the initial character(s) of the program-defined geometry names; see below for a description of these names. Using a single character (A, B, ...) will normally be best allowing the remaining seven characters (see Section 2.7.5) to be used for the rest of the geometry names.

topology — Specification of the topological I-J-K space, i.e. the number of points to generate in the three topological directions I, J and K. All geometry names generated reflect their positions in this topological space. (The names will also reflect the type of geometry.)

coordinate-system — Specification of type and position of the geometrical coordinate system to use in the geometry generation. The options are:

- CARTESIAN

- CYLINDRICAL start-xyz z-axis-xyz r-axis-xyz

- SPHERICAL start-xyz z-axis-xyz r-axis-xyz

The CARTESIAN option involves using the coordinate system of the model for the geometry generation. The CYLINDRICAL and SPHERICAL options involve defining such coordinate systems relative to the coordinate system of the model and then using these systems for the geometry generation. Note that the resulting geometry will *always* be in the cartesian coordinate system of the model.

geometry — Specification of the starting point and the vectors for mapping the topological space into the geometrical coordinate system. This will consist of:

- starting-point I-vectors J-vectors K-vectors

The starting-point determines the coordinates of the first point generated.

The number of vectors for each topological direction will correspond to the number of points defined in the topology specification, if four points are defined then three vectors are required.

The GENERATE command is somewhat complex. A few examples will show how to use it and also demonstrate its capabilities, see Figure 3.24 through Figure 3.26.

Naming convention

The GENERATE command makes use of a system for naming the geometry generated. The names of the geometrical entities will reflect both the type of geometry (point, line, etc.) and the position in the topological space. The naming system is as follows:

- Points are named:

  *id*P*ijk*

  where *ijk* is a number reflecting the position of the point in the topological space. An example:

  AP341

  here the *id* is A and the point is the 3rd in I-direction, 4th in J-direction and 1st in K-direction.

- Lines are named:

  *id*I*ijk*, *id*J*ijk*, *id*K*ijk*

  The lines in the three topological directions are identified by the letters I, J and K. *ijk* is a number taken from the one of the two points defining the line with the lowest *ijk* number.

- Surfaces are named:

  *id*S*ijk*, *id*T*ijk*, *id*U*ijk*

  The surfaces perpendicular to the three topological directions I, J and K are identified by S, T and U, respectively. *ijk* is a number taken from the one of the lines bounding the surface with the lowest *ijk* number.

- Bodies are named:

  *id*B*ijk*

  ijk is a number taken from the one of the surfaces bounding the body with the lowest *ijk* number.

- When cylindrical or spherical coordinate systems are employed such systems are implicitly defined and named:

  *id*X*ijk*

  where *ijk* = 000.

  Such a coordinate system will be the same as if the DEFINE COORDINATE-SYSTEM command created it and may be referred to by other commands (e.g. the PROPERTY command).

When ten or more points are generated in a direction two digits will be used for that topological direction in forming the name. A couple of examples: AP0341 is the name of a point being the 3rd (among 10 or more) in I-direction, 4th in J-direction and 1st in K-direction. AP030401 is the name of the same point when 10 or more points are generated for each of the three directions.

It is possible to force the names to contain two digits for any or all directions even though less than 10 points are generated. This may be desired if two digits are necessary to maintain a consistent naming system after adding more geometry later on. The ‘id’ is used to achieve this. For example, rather than only giving the character A the ‘id’ may be given as A&IIJK. This ‘id’ involves that all geometry names will contain two digits for the I-direction (because there are two I’s in the ‘id’) but only one digit for the J- and K-directions. The ‘&’ represents the character P for points, I/J/K for lines, etc. If two digits are required for all three directions then use A&IIJJKK.

The geometry names will reflect the dimension of the topology space in that only the I- and J-values will be included in the names for a 2-D topology space (and only the I-values for a 1-D space). For example, AP34 is a point in a 2-D topology space and AP3 is a point in a 1-D space.

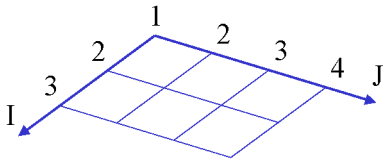
A topology range can be limited to a single value. For example, the start, end and step values: 1, 1 and 1 (the step value will in this case be irrelevant) yield the ‘range’ 1. The effect of specifying such a ‘range’ for the

K-direction rather than defining a 2-D topology space is that the K-value will be included in the geometry names. For example, a point will be named AP341 rather than AP34.

If the names constructed by this convention are already in use by previously created geometry then default names will be created.

Example 3.1 Generating a 2-D geometry

The command below generates a 2-D geometry consisting of surfaces, lines and points. There will be 3 points in I-direction and 4 points in J-direction. (n_i and n_j represent the number of elements to divide the lines into in a subsequent meshing.) The topological space is illustrated to the right. The resulting geometry is displayed below. Notice the system for naming the geometric entities.



topology: 1 to 3 with step 1 in I-direction, 1 to 4 with step 1 in J-direction, no specification for K-direction

```
GENERATE SURFACE A  1 3 1 ni  1 4 1 nj END
CARTESIAN  0. 0. 0.                          ← the starting-point is in the origin
1. 0.0 0.  1.6 0.0 0.  END                   ← the two vectors in I-direction
0. 0.3 0.  0.0 0.7 0.  0. 1.1 0.  END        ← the three vectors in J-direction
```

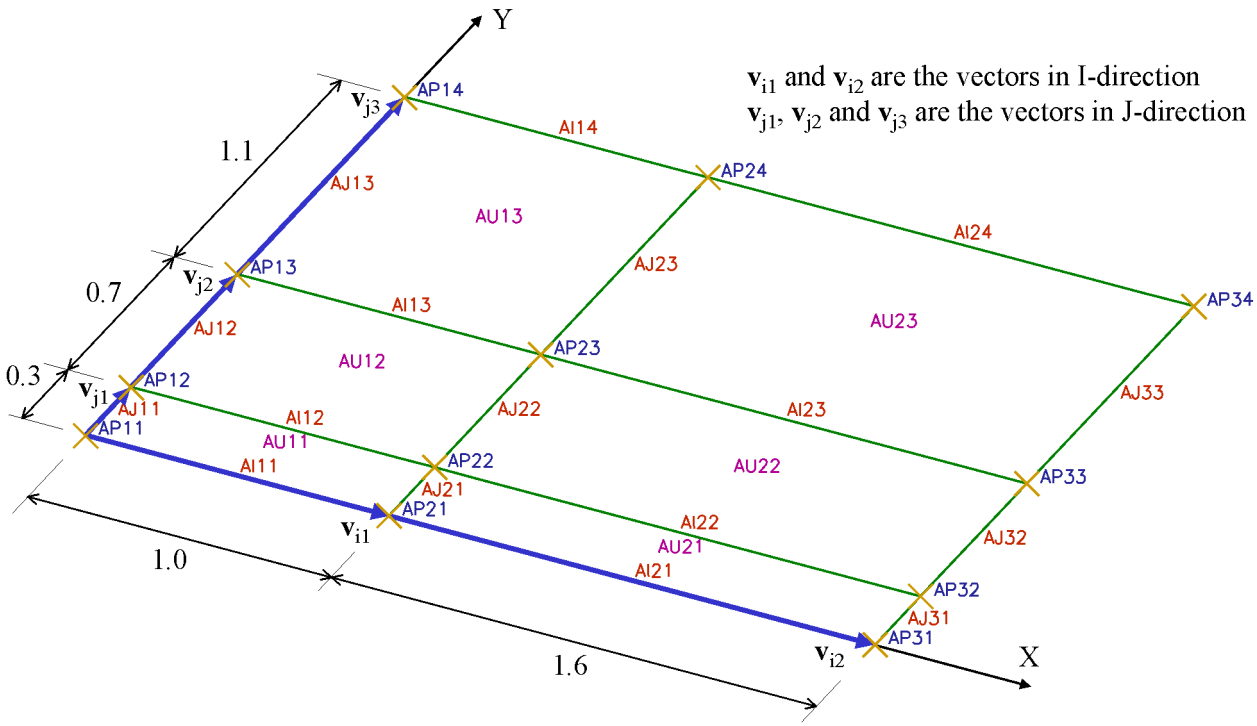
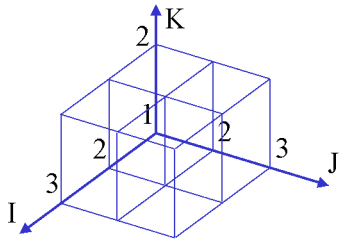


Figure 3.24 Generation of a 2-D geometry

Example 3.2 Generating a 3-D geometry

The command below generates a 3-D geometry consisting of lines and points. There will be 3 points in I- and J-directions and 2 points in K-direction. (n_i, n_j and n_k represent the number of elements to divide the lines into in a subsequent meshing.) The topological space is illustrated to the right. The resulting geometry is displayed below. (Only point names are displayed.)



topology: 1 to 3 with step 1 in I, 1 to 3 with step 1 in J, 1 to 2 in K

```
GENERATE LINE A  1 3 1 ni   1 3 1 nj   1 2 1 nk
CARTESIAN  0. 0. 0.                          ← the starting-point is in the origin
3. 0. 0.0  2. 0.7 0.3  END                   ← the two vectors in I-direction
0. 1. 0.0  0. 2.0 0.0  END                   ← the two vectors in J-direction
0. 0. 2.5  END                               ← the single vector in K-direction
```

$\mathbf{v}_{i1}$ and $\mathbf{v}_{i2}$ are the vectors in I-direction
$\mathbf{v}_{j1}$, and $\mathbf{v}_{j2}$ are the vectors in J-direction
$\mathbf{v}_{k1}$ is the vector in K-direction

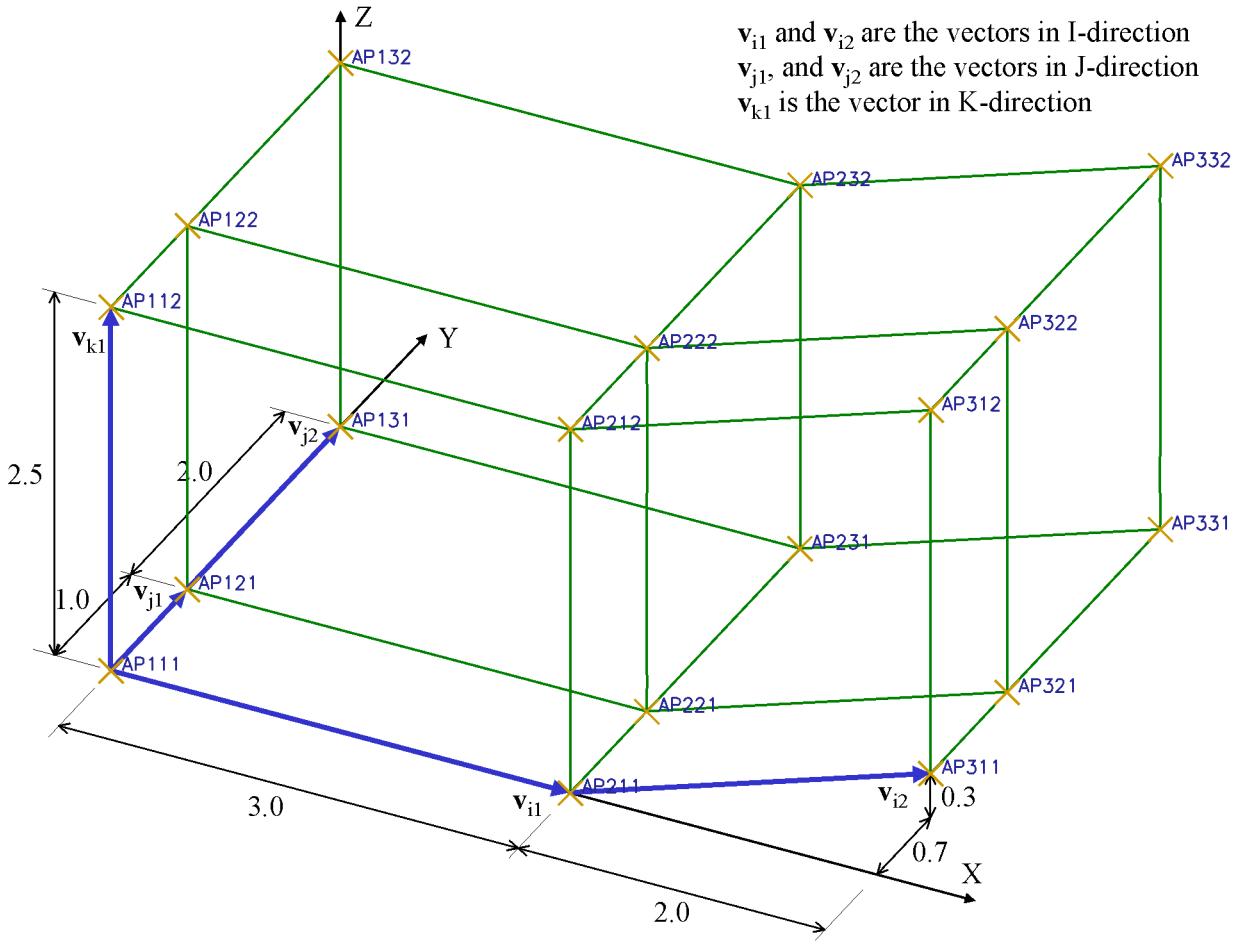
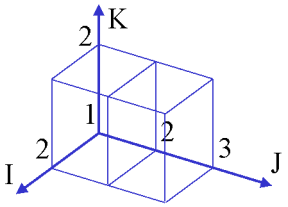


Figure 3.25 Generation of a 3-D geometry

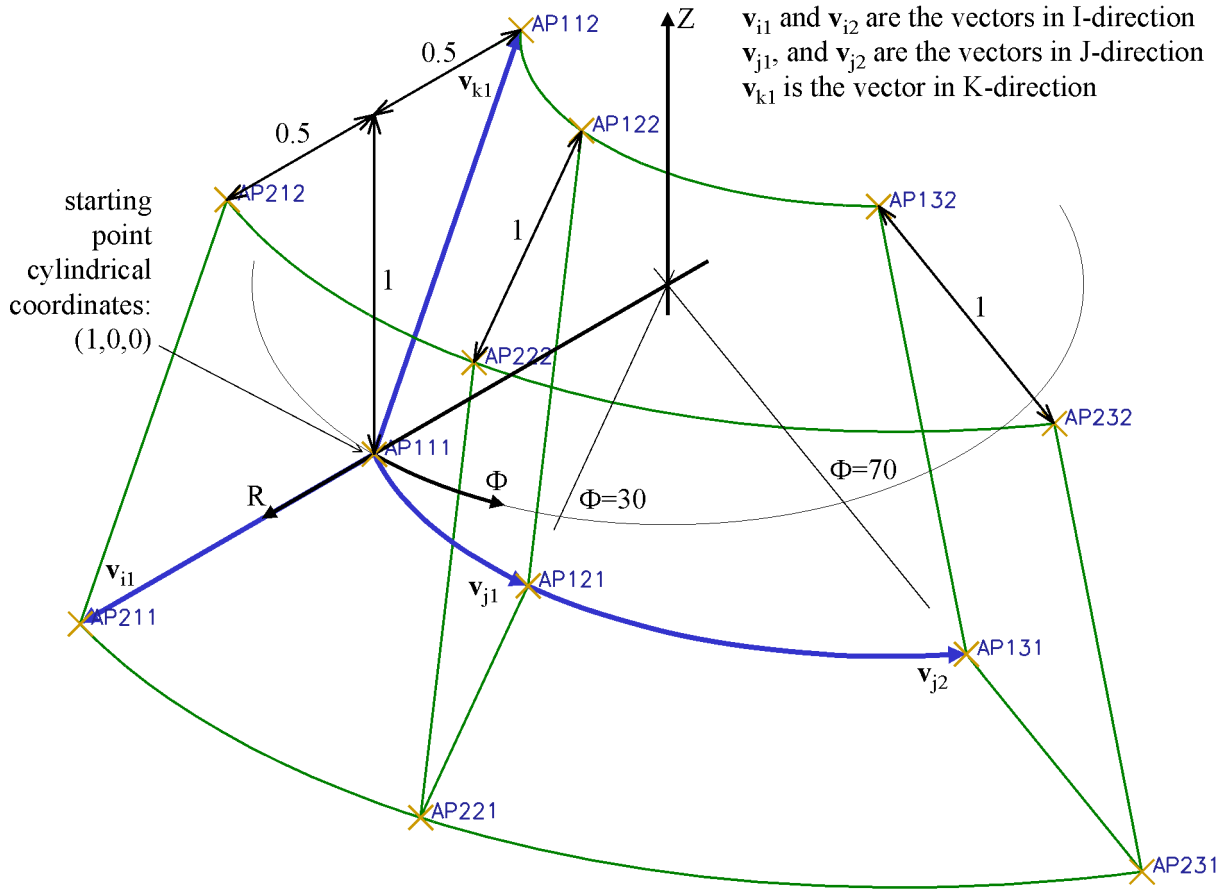
Example 3.3 Generation of a 3-D geometry in cylindrical coordinate system

The command below generates a 3-D geometry consisting of bodies, surfaces, lines and points. There will be 2 points in I-direction, 3 points in J-direction and 2 points in K-direction. (n_i, n_j and n_k represent the number of elements to divide the lines into in a subsequent meshing.) The topological space is illustrated to the right. Note that the vectors are given in the cylindrical coordinate system R, Φ, Z. The resulting geometry is displayed below. (Only point names are displayed.)



topology: 1 to 2 in I, 1 to 3 with step 1 in J, 1 to 2 in K

```
GENERATE BODY A  1 2 1 ni  1 3 1 nj  1 2 1 nk
CYLINDRICAL  0. 0. 0.  0. 0. 1.  1. 0. 0.   ← the coordinate system
1. 0. 0.                                    ← the starting-point is in R=1, Φ=0, Z=0
 1.0  0. 0.  END                            ← the single vector in I-direction
 0.3 30. 0.  0.4 40.0 0.0  END              ← the two vectors in J-direction
-0.5  0. 1.  END                            ← the single vector in K-direction
```



v_{i1} and v_{i2} are the vectors in I-direction
v_{j1}, and v_{j2} are the vectors in J-direction
v_{k1} is the vector in K-direction

Figure 3.26 Generation of a 3-D geometry in cylindrical coordinate system

3.3.8 Extruding Geometry

See Section 3.3.9 for a brief discussion on the EXTRUDE command versus the GENERATE command.

The EXTRUDE command is typically used for creating a 3-D geometry model by extruding a 2-D geometry model, for example extruding a plane surface to form a box-like model. The command is, however, general and is capable of extruding and copying any type of geometry (for example point to line or arc, line to plane or cylinder). The extrusion may be done in the global cartesian coordinate system or in a cylindrical coordinate system defined within the EXTRUDE command.

The EXTRUDE command actually performs a repetitive combined copying and extrusion as illustrated in Figure 3.27. The original geometry is copied and extruded along vectors. Notice that the created geometry is the same as the one used to illustrate the GENERATE command in Figure 3.23.

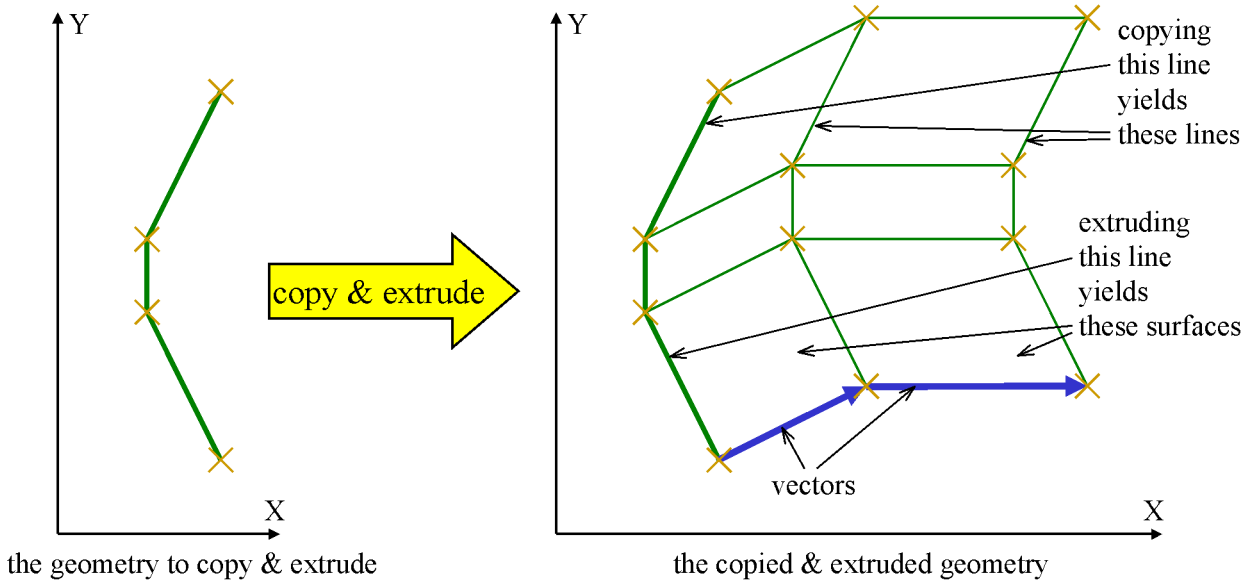


Figure 3.27 The EXTRUDE command copies and extrudes an original along vectors

As is the case for the GENERATE command a single EXTRUDE command will typically replace a number of commands for defining points, lines, surfaces and (if relevant) bodies, i.e. a number of DEFINE POINT ..., DEFINE LINE ..., DEFINE SURFACE ... and DEFINE BODY ... commands. The structure of the command is as follows:

EXTRUDE copy-geometry extrude-geometry coordinate-system copy-extrude-vectors extrude-what

The parameter names above each represent several parameters as detailed in the following. The EXTRUDE command is described in principle below; for a complete description see the command description in Chapter 5.

copy-geometry	Select geometry to copy. A prefix for the geometry names of the copies is given to enable identifying the copies later on.

extrude-geometry	Select geometry to extrude. Points are extruded to lines, lines to surfaces and surfaces to bodies. A prefix for the names of the extruded geometry is given to enable identifying this later on.
coordinate-system	Select the global cartesian or a specified cylindrical coordinate system for the copy/extrusion operation.
copy-extrude-vectors	Give the number of copies/extrusions to make. The corresponding number of vectors are given one by one or several at a time.
extrude-what	Choose whether and what to extrude: copy-only (do not extrude anything), extrude points to lines, extrude lines to surfaces, extrude surfaces to bodies.

3.3.9 The GENERATE Command versus the EXTRUDE Command

The GENERATE and EXTRUDE commands have overlapping application areas (several types of models may be created by either command). They differ, however, in that EXTRUDE involves a bottom-up while GENERATE more of a top-down approach to modelling. This different approach to modelling may be explained as follows:

- In order to take full advantage of the GENERATE command you basically first need to decide what the desired geometry model should look like, i.e. where all points, lines and surfaces should be. Refining and adjusting the geometry model after giving the command is highly relevant but the major part of the geometry should be in place. Used in this way the GENERATE command is extremely powerful. The examples of Figure 3.24 through Figure 3.26 illustrate this top-down approach to modelling.

- The EXTRUDE command is based on first creating a typical section of the desired geometry model and thereafter extruding this to form a more complete model. I.e. you start by creating a simplified model and then build on this one to establish the complete model. The tutorial example of Section 3.2 illustrates this bottom-up approach to modelling.

3.3.10 Importing DXF File

The READ DXF command will read a DXF-formatted file. DXF is a format for exchanging 3D CAD data (DXF originates from the CAD program AutoCAD). Based on the information in the file Prefem will establish a geometry model. This model may be extended and modified.

A DXF-file may contain several types of information of which Prefem will interpret a selection as described in Chapter 5.

3.3.11 Geometry Names

Adopting a systematic convention for naming the geometry will ease both modelling and later interpretation of the geometry model. The user is advised to decide a naming convention prior to commencing the modelling. The naming convention employed by the GENERATE command, see Section 3.3.7, may be used as a guide. Also refer to Section 3.9 for a discussion on how a systematic convention for naming the geometry may be utilised for easy reference to several geometrical entities.

3.4 Creating the FE Model

Having defined the geometry, plus additional data as described below, the FE mesh is automatically created by the MESH command.

Finite elements may only be created on the appropriate type of geometry, i.e. beam elements may only be created where there are geometry lines, shell elements only where there are surfaces, etc. Figure 3.28 illustrates this principle.

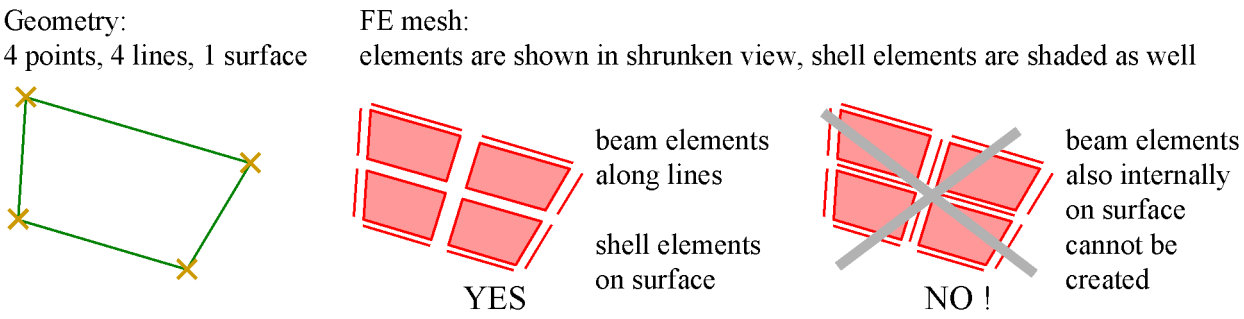


Figure 3.28 Finite elements are only created on appropriate geometry

The meshing is performed in the following sequence:

- Beam or truss elements are created for lines/curves, if relevant. If such elements are not requested then only discretisation data for determining the surface mesh is established.
- Membrane or shell elements are created for surfaces, if relevant. If such elements are not requested then only discretisation data for determining the body mesh is established.
- Solid elements are created for bodies, if relevant.

It follows from the above that making a surface mesh (e.g. shell elements) with and without line elements (e.g. stiffener beams) is governed by whether beam elements have been requested in addition to shell elements.

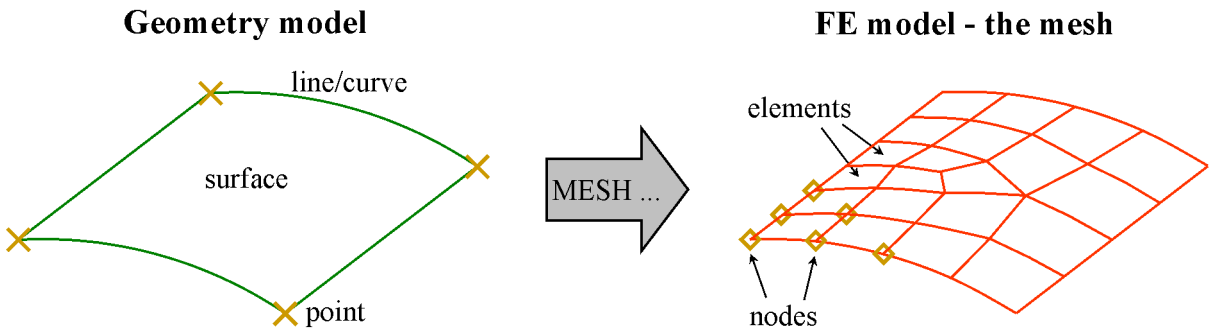


Figure 3.29 The MESH command automatically creates the FE mesh based on a geometry model

3.4.1 Controlling the Mesh Creation

The following data must be defined to enable creation of the FE model:

- Data determining the element discretisation:
  - — The number of elements is determined when defining lines and curves. This setting is changed by the commands SET NUMBEROF-ELEMENTS and SET MAX-ELEMENT-LENGTH.
  - — The spacing of elements for lines and curves is by default even, i.e. all elements (element edges) are equal in length. The command SET ELEMENT-LENGTH-RATIO is used to set the ratio between the element lengths, see Figure 3.30.
- Desired type(s) of element is set by the SET ELEMENT-TYPE command. See Section 2.5 for information on types of elements available.

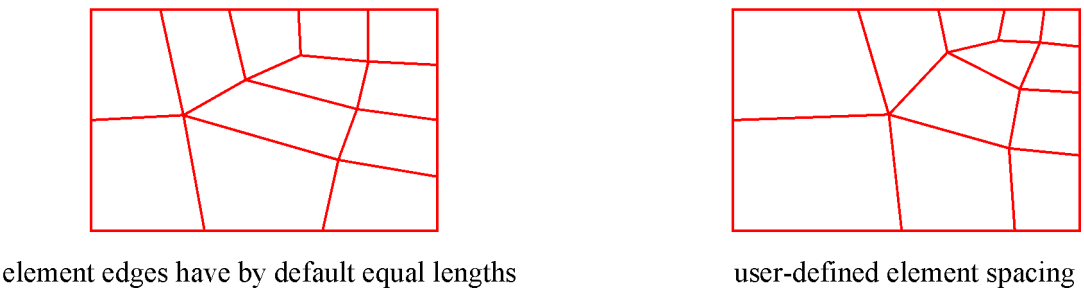


Figure 3.30 Example of default and user-defined element spacing

A maximum of four mesh-corners are allowed for a surface; see Figure 3.31. There is one exception: when there is only one element between two mesh-corners in which case five mesh-corners are allowed; see Figure 3.39. The number of mesh-corners is set by the SET MESH CORNER-TYPE command.

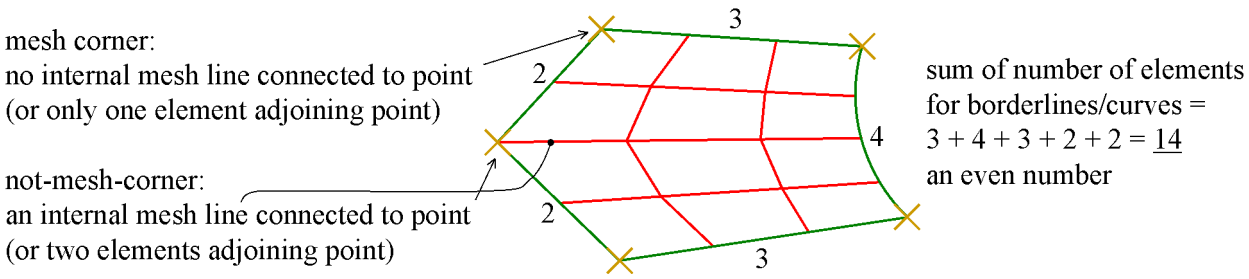


Figure 3.31 Requirements for creating a surface mesh of quadrilateral elements

When creating a surface mesh of quadrilateral elements Prefem will attempt to minimise the number of elements for each surface and at the same time seek to make the quadrilateral elements as square as possible.

If you want to avoid the inferior triangular element then make sure the sum of the number of elements for the lines/curves surrounding the surface is an even number. If the sum is odd then a triangular element will

be inserted in the sharpest corner (or you may manually set in which corner there should be a triangular element). For an illustration of this see Figure 2.4.

Adjoining surfaces are defined by common lines/curves and adjoining bodies are defined by common surfaces. As the meshing (discretisation) sequence is: first lines/curves, then surfaces and at last bodies it follows that only one set of nodes is created for the common lines/curves for surfaces and common surfaces for bodies. Merging nodes created for different surfaces and bodies is therefore not required.

Note: You are allowed to define overlapping lines/curves. If two neighbouring surfaces are defined with such overlapping lines/curves rather than sharing a common line/curve then a double set of nodes will be created involving that the surfaces will not be coupled, there will be a crack in the model. See Figure 3.32 for an illustration of this. The same goes for neighbouring bodies defined with overlapping surfaces. There will be overlapping lines, curves and surfaces only when defined on purpose. Neither of the features for automatic creation of geometry (COPY, GENERATE, EXTRUDE, etc.) will result in overlapping lines, curves and surfaces.

Note: If you are in doubt about whether you have double nodes in your model, caused by overlapping geometry or other, then use the CHECK CLUSTERED-NODES command to identify closely located nodes. If such is undesired the mesh, and possibly the geometry, should be revised.

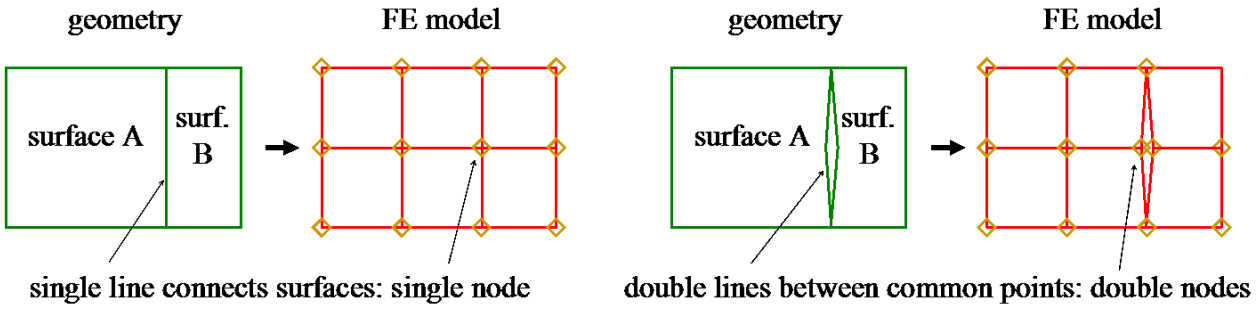


Figure 3.32 Adjoining surfaces are defined by common lines

Through the CHECK ELEMENT-SHAPE command the shape of the elements may be checked. This is relevant for surface elements (membranes and shells) and for volume elements (solids).

An unsatisfactory mesh may be deleted by the DELETE MESH command. You may then modify the data determining the element discretisation and create a new mesh.

Note: For large models the MESH command may require some time. During processing of the command you may use Shift+Esc to abort it. This will have no effect on the program execution (apart from incomplete execution of the command in question) and you may continue modelling. This is available on PC only; a similar feature is not available on other operating systems.

The following factors will determine the mesh to be created:

- The geometry of the lines, surfaces and bodies to be meshed
- The direction of the definition of lines and surfaces

- In some cases the start and finishing point of a surface
- The number and relative size of the elements along the lines/curves of the surfaces
- The definition of mesh corner types of the surfaces (use command SET MESH CORNER-TYPE)
- The setting of the MESH-PARAMETER (use command SET MESH-PARAMETER)

Note: Beam and shell elements have 6 d.o.f.s while solid elements have 3 d.o.f.s. These element types may be used in the same model and have common nodes provided that the beam and shell elements are created before the solid elements. This ensures that the nodes get 6 d.o.f.s. The MESH ALL command follows this meshing sequence. If you create mesh on parts one by one then you must see to yourself that the meshing sequence is correct.

Note: Truss elements have 3 d.o.f.s. This element type cannot be used as stiffener for shell elements. The reason for this is: In order to create a surface mesh (shell elements) the bounding lines must be meshed first. If these are meshed with truss elements the number of d.o.f.s will be set to 3. Prefem is unable to increase number of d.o.f.s for existing nodes from 3 to 6 which would be required to add shell elements.

3.4.2 Elements for Points

Certain elements are defined directly for geometry points. These are the spring, damper and mass elements; see Table 2.2 and Figure 2.3. The modelling of these is somewhat different from the other elements in that the DEFINE command rather than the SET ELEMENT-TYPE command is used to define their existence. Note that their material must have been defined using the PROPERTY MATERIAL command *prior* to defining them. The elements are created by the MESH ALL or MESH element-name commands (element-name is the name established in the DEFINE command).

Note: Spring (axial and to-ground), damper (axial and to-ground) and mass elements can *only* be defined between/connected to geometry points.

3.4.3 1-D Elements for Lines/Curves

The 1-D elements are the straight 2 node truss and beam elements and the curved 3 node beam element; see Table 2.2 and Figure 2.3. Note that for 1-D elements created on curves (arcs, splines, etc.) only the nodes will lie on the curve; see Figure 3.33.

Figure 3.33 Straight 2 node truss and beam elements and curved 3 node beam element

3.4.4 2-D Elements for Surfaces

The 2-D elements are the membrane, shell (including sandwich and layered) and axi-symmetric elements; see Table 2.2 and Figure 2.3.

Figure 3.34 through Figure 3.44 illustrate various types of mesh that may be created for surfaces. A brief description is given below for each figure. See also Figure 3.45 with corresponding note.

- Figure 3.34 illustrates how both quadrilateral and triangular elements may be used to create a mesh for a surface with four mesh-corners. The four borderlines are divided into 3, 3, 4 and 4 equally sized element edges. Note that the mesh with triangular elements is created by first making a mesh with quadrilateral elements and then splitting each quadrilateral element into two triangular elements.

- Figure 3.35 illustrates an alternative mesh for the same surface as in Figure 3.34; the only difference is that the number of mesh-corners has been reduced to three. Observe how the mesh is influenced by the number of mesh-corners.

- Figure 3.36 illustrates a mesh for the same surface, now with only two mesh-corners. In this case the mesh becomes very distorted so for a quadratic surface this choice of mesh-corners combined with discretisation data is obviously a bad one.

- Figure 3.37 illustrates a mesh for the same surface with only one mesh-corner.

- And Figure 3.38 illustrate a mesh for the same surface with no mesh-corners.

- Figure 3.39 shows a special case for which five mesh-corners are allowed; there can be only one element edge in-between two of the mesh-corners.

- Figure 3.40 shows a radial mesh with triangular elements. This mesh, which is suitable in the middle of a sector shaped geometry, is created when the following conditions are met:
  — The triangular surface has three mesh-corners
  — Two of the lines are discretisised into only one element edge, the third line/curve (which may also be several lines/curves with not-mesh-corner in-between) may have any number of elements
  — Triangular elements are requested

- Figure 3.41 shows an alternative mesh for a sector shaped geometry. This mesh is created when the following conditions are met:
  — Define the centre of the sector as a *cut-corner* (use the command SET MESH CORNER-TYPE ... CUT-CORNER).
  — Let the two lines joining in the cut-corner have equal number of elements.

- Figure 3.42 shows more alternative meshes for a sector shaped geometry. Comparing the left most of the three meshes with the mesh of Figure 3.41 it is seen that the only difference is that the number of elements for the arc has been changed from 6 to 7. Since the sum of elements around the surface is now an odd number a triangular element is inserted in the cut-corner. The other two alternative meshes illustrate the so-called small-cut-corner and large-cut-corner. Observe how the mesh changes, which is the better one depends on the geometry, e.g. how narrow the sector is.

- Figure 3.43 shows alternatives for meshes with a triangular element in a (sharp) corner. Rather than studying the logic of the options the figure may be used as follows: Decide which mesh you want, find which option this corresponds to and use the SET MESH CORNER-TYPE command to specify that option.
- Figure 3.44 shows how the SET MESH EDGE RECTANGULAR command is used to solve the particular problem of meshing a surface with a notch (not necessarily 90 degrees).

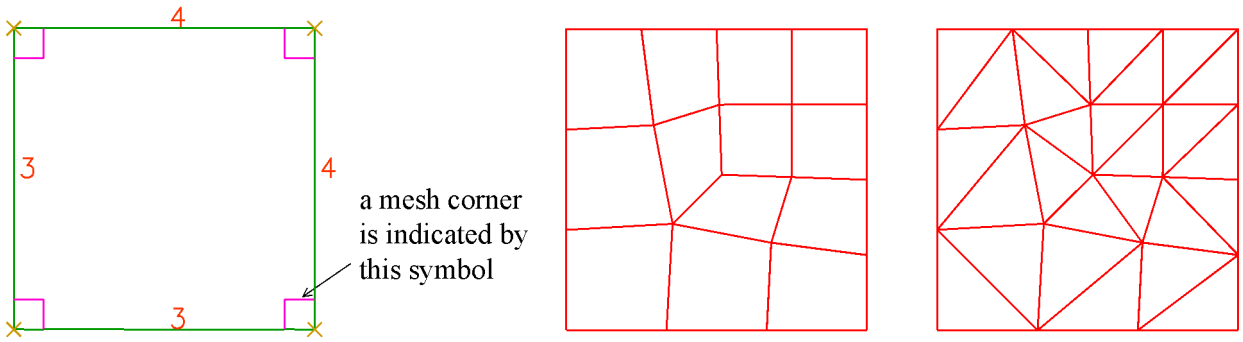


Figure 3.34 Quadrilateral and triangular element mesh for surface with 4 mesh-corners

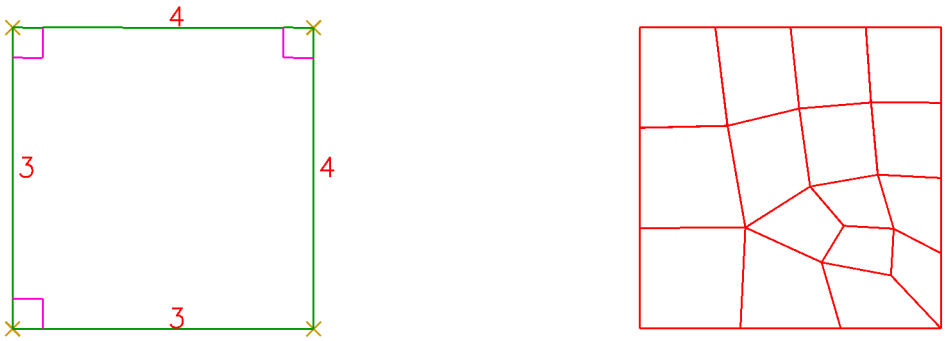


Figure 3.35 Quadrilateral element mesh for surface with 3 mesh-corners

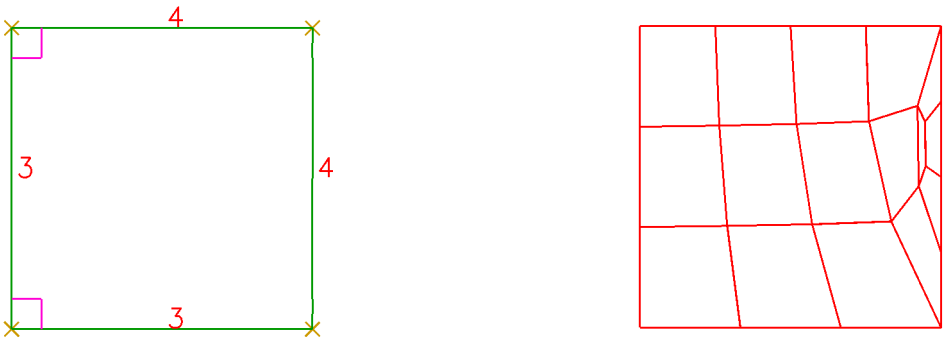


Figure 3.36 Quadrilateral element mesh for surface with 2 mesh-corners — a bad mesh in this case

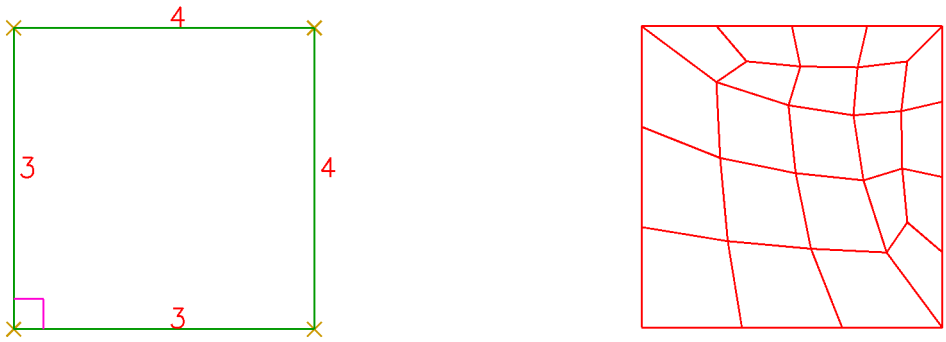


Figure 3.37 Quadrilateral element mesh for surface with 1 mesh-corner

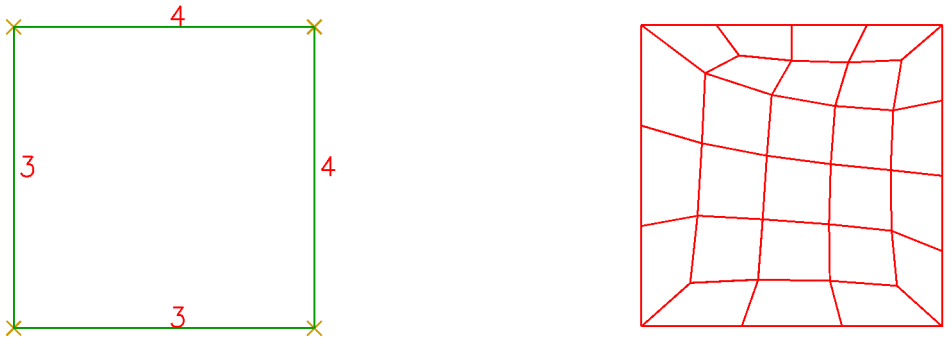


Figure 3.38 Quadrilateral element mesh for surface with no mesh-corners

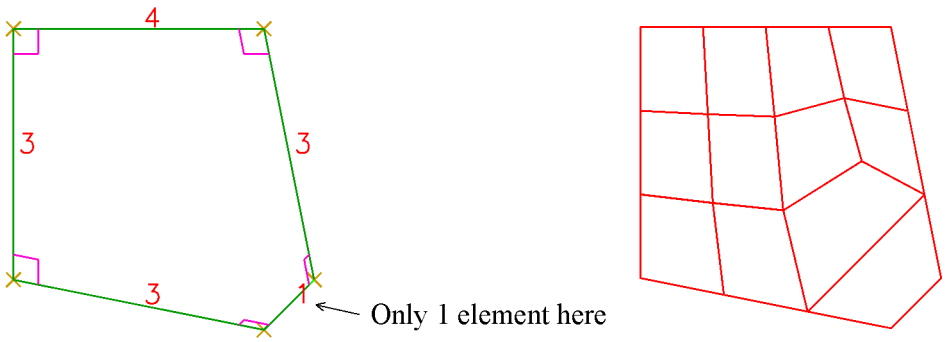


Figure 3.39 Quadrilateral element mesh for surface with 5 mesh-corners — a special case

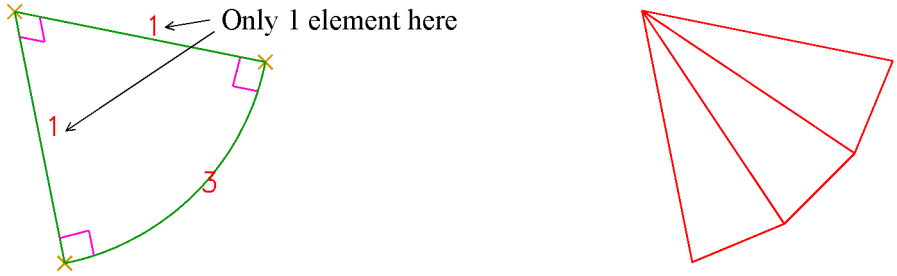


Figure 3.40 Radial triangular element mesh for surface with 3 mesh-corners — a special case

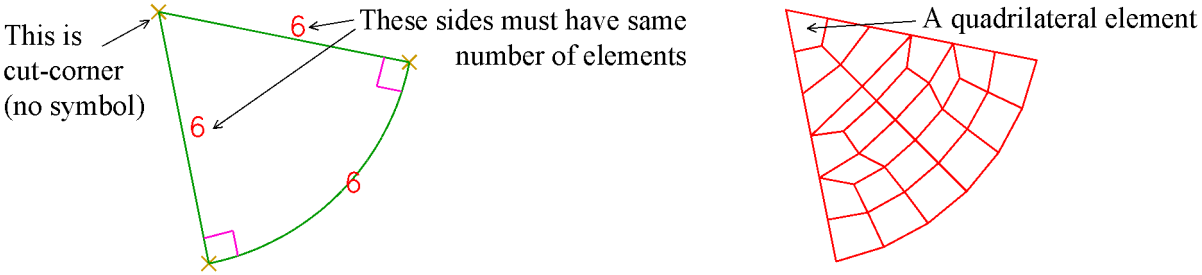


Figure 3.41 Quadrilateral element mesh for surface with 2 mesh-corners and a cut-corner, even sum of elements

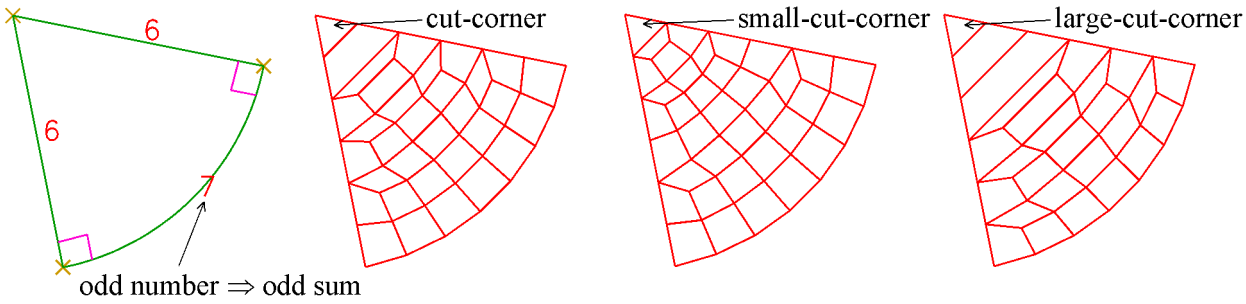


Figure 3.42 Quadrilateral element mesh for surface with 2 mesh-corners and a cut-corner, odd sum of elements gives a triangular element in cut-corner, small-/large-cut-corners yield different meshes

- Even sum around surface:
  1. Only quadrilaterals - 'normal mesh'
- If odd sum around surface then triangular element in sharpest corner
- Demand triangular element in selected point (sharp corner) by:
  TRIANGULAR-ELEMENT
- Choose alternative transitions from triangular element to mesh in rest of surface (numbers refer to illustrations):
  2. TRIANGULAR-PRE-NONE
  3. TRIANGULAR-POST-NONE
  4. TRIANGULAR-BOTH-NONE
  5. TRIANGULAR-PRE-ONE
  6. TRIANGULAR-POST-ONE
  7. TRIANGULAR-PRE-TWO
  8. TRIANGULAR-POST-TWO

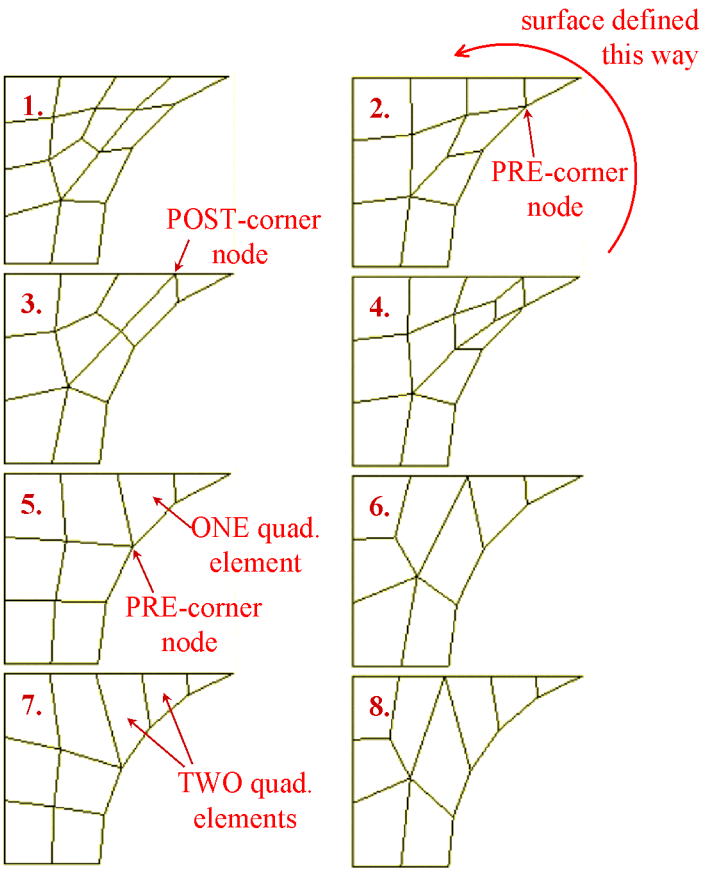


Figure 3.43 Alternatives for triangular element in corner

Use command:
SET MESH EDGE RECTANGULAR
surface line number-of-elements
to get mesh as shown

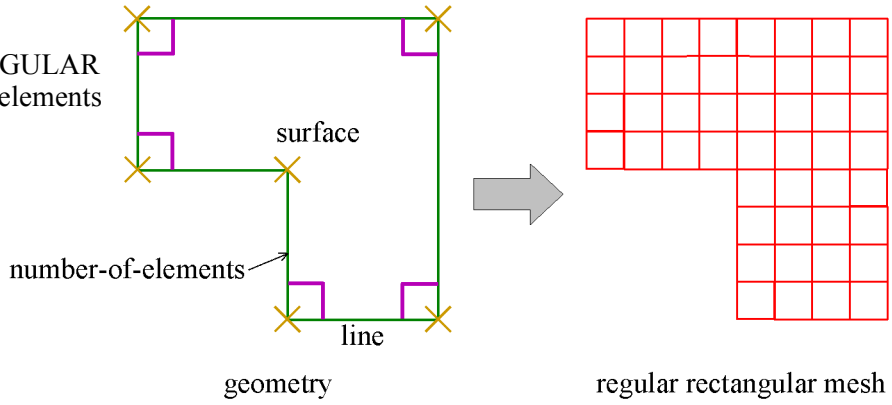


Figure 3.44 Mesh for an L-shaped surface

Note: The mesh created for a surface with dent or concavity tends to be distorted. You may need to cut the surface so as to split the dent to get an acceptable mesh. See Figure 3.45.

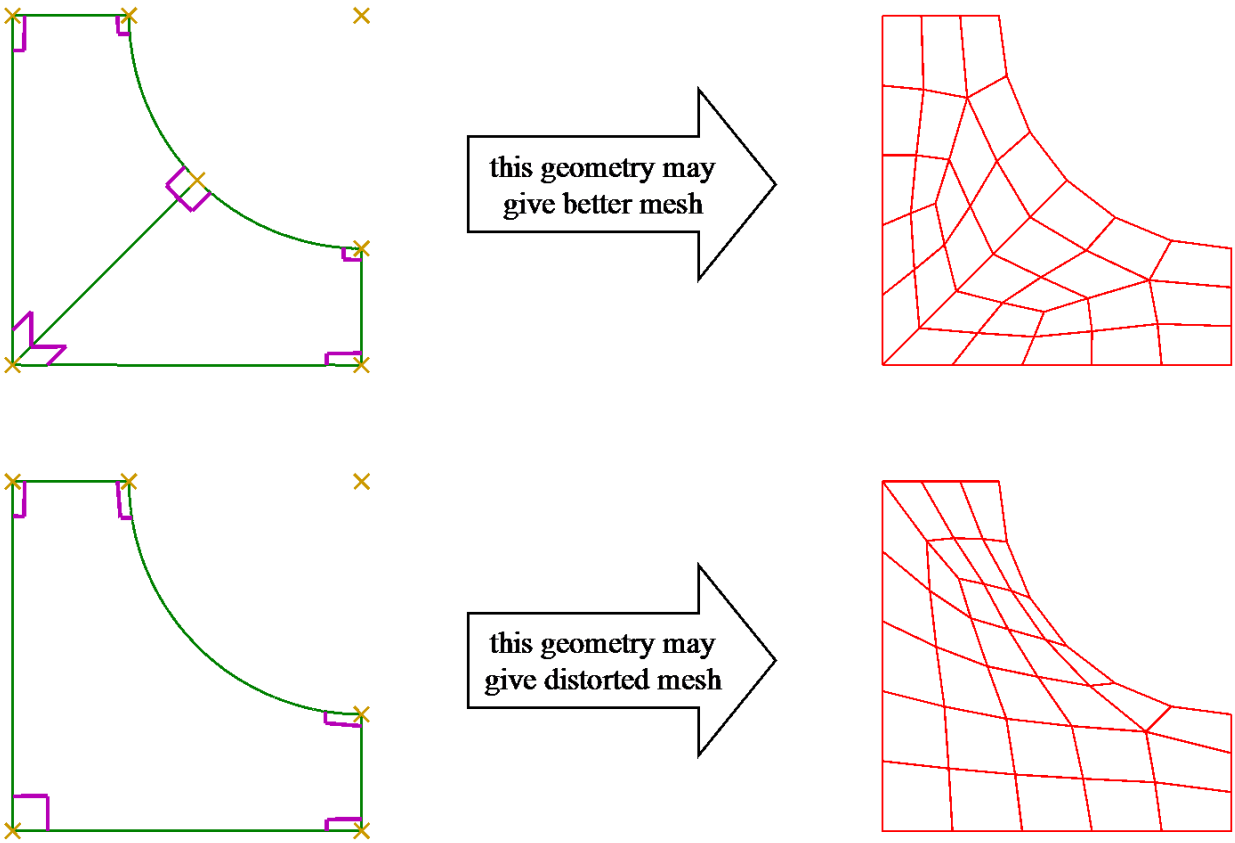


Figure 3.45 Mesh for a surface with dent or concavity

3.4.5 2-D Elements for Curved Surfaces

The previous section explained various types of 2-D element meshes for *plane* surfaces. This section deals with meshes for *curved* surfaces. Unless all its borderlines/curves lie in a plane a surface will be curved.

Generally, the interior of a surface is not defined explicitly unless projected onto a shape. Even in cases where all borderlines/curves do lie in a plane the interior *may* be projected onto a surface as shown in Figure 3.46.

Surfaces with borderlines/curves forming a cylinder/sphere will in certain cases yield cylindrical/spherical element meshes even without projecting the surfaces onto cylindrical/spherical shapes. These cases are:

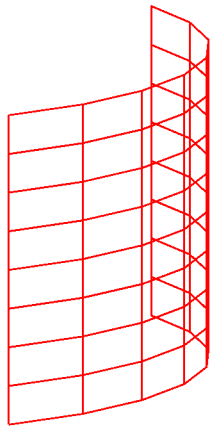
- When a regular mesh is created, i.e. equal number of elements on opposite edges
- When regular or irregular meshes are created using a cylindrical/spherical coordinate system. (The iteration process creating the mesh will then be performed in the cylindrical/spherical coordinate system.) This will be done:

— For surfaces defined by the GENERATE command in a cylindrical/spherical coordinate system (except when the surfaces become triangular instead of quadrilateral; see the dome sector of Figure 3.17, the mesh must be projected onto a spherical shape even when the GENERATE command has been used)

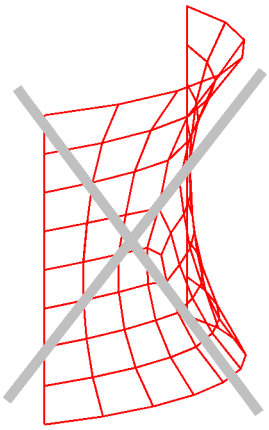
— For surfaces for which a cylindrical/spherical coordinate system has been set by the SET MESH command

Figure 3.46 illustrates the effect of projecting and not projecting various meshes onto shapes.

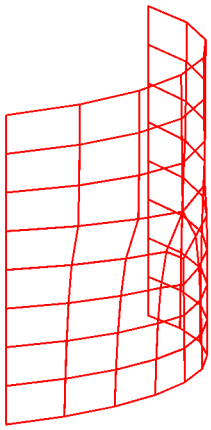
Generally, the mesh for curved surfaces will have a tendency to 'sink in'. This because the iteration process creating the mesh seeks to minimise the 'potential energy' of the surface. Note that the nodes on the borderlines/curves will not be projected. The borderlines/curves of a surface should therefore lie in the shape onto which the surface is projected, otherwise the elements on the edge of the surface will be distorted.



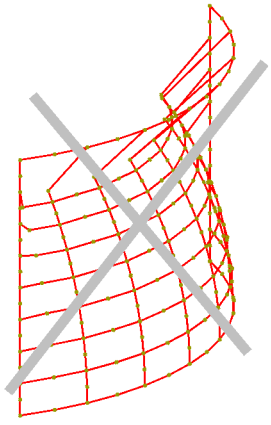
Regular meshes for cylinders need not be projected.



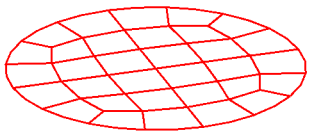
Irregular meshes will 'sink in' if not projected.



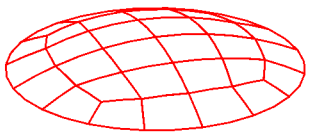
Irregular meshes should be projected onto shapes, in this case a cylinder. Alternatively, the mesh may be created in a cylindrical/spherical coordinate system.



Projection onto a sphere of a mesh with borderlines/curves on a cylinder will give distorted elements.



Mesh for a circular plane surface will be plane.



Mesh for a circular plane surface projected onto a sphere. The border-curves of lie in the spherical shape.

Figure 3.46 Projection of element meshes onto shapes

3.4.6 3-D Elements for Bodies

The 3-D elements are the solid elements; see Table 2.2. A solid element mesh must be prismatic, i.e. the meshes of the top and bottom surfaces must in terms of topology be equal. This means that their corresponding borderlines/curves must have equal number of elements and they must have mesh-corner and not-mesh-corner in corresponding points. See Figure 3.47.

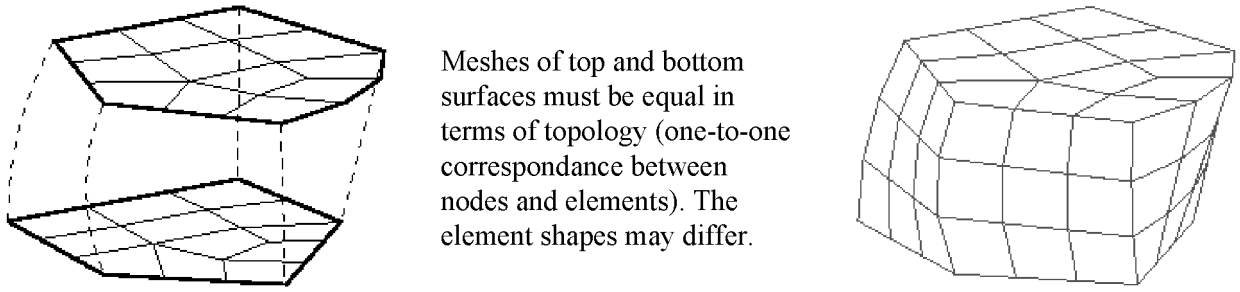

Meshes of top and bottom surfaces must be equal in terms of topology (one-to-one correspondance between nodes and elements). The element shapes may differ.

Figure 3.47 Mesh for solid elements must be prismatic

3.4.7 Changing the Mesh Created

The mesh created may be altered by changing the element discretisation, i.e. the number of elements along lines/curves, element sizes and the relative sizes of elements along lines/curves. Nodal coordinates may also be changed even though this should normally be avoided.

Changing the number of elements (SET NUMBEROF-ELEMENTS) and element sizes (SET MAX-ELEMENT-LENGTH) will result in deletion of any current mesh and re-creation of a new mesh (if possible). The new mesh will automatically be displayed (provided that there was a mesh before the change and that this was displayed). Prior to changing relative sizes of elements along lines/curves (SET ELEMENT-LENGTH-RATION), however, any current mesh must be deleted (DELETE MESH ALL).

The command CHANGE NODE allows giving new cartesian coordinates for a node. Note, however, that this modification will be lost once the mesh is deleted and a new mesh created.

The MESH ADJUST command will change the element discretisation so that all surfaces can be meshed.

The SET DEFAULT ADJUST-MESH command (or pressing the Shortcut command ‘Mesh adj’) sets a mode in which the MESH ADJUST command is automatically executed whenever the element discretisation is changed.

3.5 Defining and Assigning (Connecting) Properties

Material data, beam cross sections, plate thicknesses, boundary conditions, loads, etc. are so-called properties that are assigned or connected to the relevant geometry by referring to the appropriate geometry names. The properties will automatically be transferred to the FE model. The command for defining properties is PROPERTY. The various properties are discussed in the subsections below. The property types ‘load’, ‘initial-displacement’ and ‘initial-velocity’, however, are discussed in Section 3.6. And the property type ‘transformation’ is discussed in Section 3.11.1.

Within the PROPERTY command there is a need for reference to geometrical entities by their names. Refer to section Section 3.9 for a discussion on how to do this.

It is possible to assign certain properties directly to nodes (e.g. PROPERTY BOUNDARY-CONDITION and PROPERTY LINEAR-DEPENDENCY). Note that these will be deleted if the mesh is deleted and re-created. Therefore, if properties need to be assigned to nodes and elements it is advisable to assign such properties at the very end of the modelling.

Properties are changed by either the CHANGE PROPERTY or the PROPERTY CHANGE command, these commands are equivalent. Properties are deleted by the DELETE PROPERTY command.

3.5.1 Beam Cross Section

The following cross sections are available and may be assigned to the truss element and the two and three node beam elements (see Appendix B for illustrations of the cross sections):

- Bar
- Box
- Channel
- Double bottom
- General
- I
- L
- Pipe
- Un-symmetrical I

The sections are defined by the PROPERTY SECTION command and given unique names. The cross sections are then assigned to the appropriate lines/curves for which truss or beam elements are created. This assignment is performed by the CONNECT SECTION command by referring to the section names and geometry line/curve names. Assigning sections to lines/curves for which no truss or beam elements are created has no consequence.

Note: A T-section may be created based on an I-section as explained in Figure 3.48.

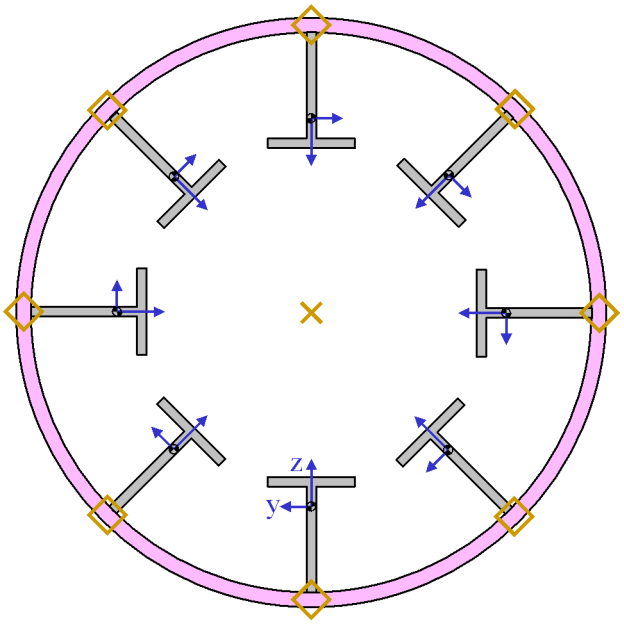
3.5.2 Local Coordinate System for Beam

Local coordinate systems must be defined for beam elements in order to orientate their cross sections. The PROPERTY LOCAL-COORDINATE-BEAM command is used to assign a definition to the appropriate lines for which beam elements are created.

The local x-axis will always follow the neutral axis of the beam element and pointing in the direction from the first to the second (or next) geometry point used in the definition of the line/curve. Orientating the local coordinate system therefore reduces to orientating either the local y- or z-axis which again is a matter of ori-

entating the local y-x-plane or z-x-plane respectively. As both planes go through the neutral axis (= the local x-axis) of the beam element a single point not positioned on the neutral axis will determine the complete local coordinate system. This single point or guiding point may be defined in various ways as explained for the command in Chapter 5.

Figure 3.48 shows an example of how the PROPERTY LOCAL-COORDINATE-BEAM is used in combination with the PROPERTY ECCENTRICITY command (see Section 3.5.3) to make the internal stiffeners of a cylindrical tank point towards the centre of the cylinder and be connected with eccentricities to the nodes (the stiffeners are welded onto the inner tank wall).



The orientation of the T-stiffeners of this model is achieved by the following commands:

```
PROPERTY LOCAL-COORDINATE-BEAM
      ZX-PLANE TOWARDS-FIXED-POINT
      coordinates-of-centre-point select-lines

PROPERTY ECCENTRICITY-BEAM
       select-lines
       CALCULATED-POSITIVE-Z-OFFSET
```

Note that a T-section is created based on an I-section by making the width of the flange (in this case the bottom flange) a fraction larger than the web thickness. Give the thickness of the same flange a 'reasonable value'.

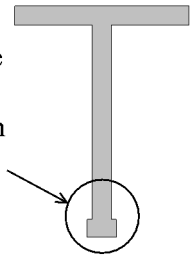


Figure 3.48 Local coordinate system and eccentricities for internal stiffeners in tank

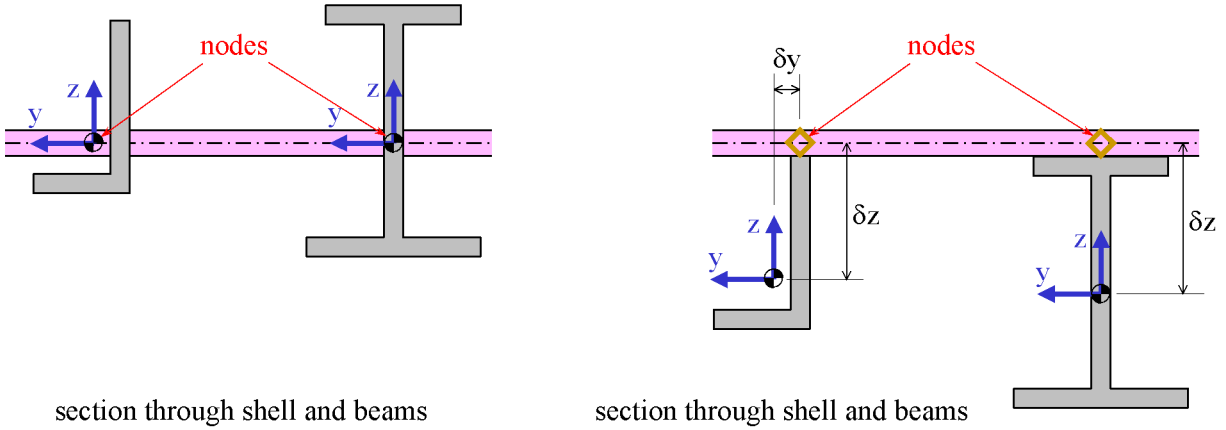
If the local coordinate system is not defined manually an automatic definition will take effect as explained for the command.

For truss elements the local coordinate system has no consequence and may be neglected.

3.5.3 Eccentricity for Beam

Two and three node beams may have offsets or eccentric attachments to the nodes. The PROPERTY ECCENTRICITY command defines such eccentricities for selected lines, either as vectors from the nodes to the beam element ends (and midpoint for 3 node beam), or by an automatic feature in which the program determines the eccentricity required to let the beam section be welded or attached onto the surface of the plate or shell. The automatic feature, termed CALCULATED-NEGATIVE-Z-OFFSET and CALCULATED-POSITIVE-Z-OFFSET in the command, takes the sectional data and plate/shell thickness into account, these data therefore need to be defined prior to requesting the automatically calculated eccentricity.

The eccentricity is constant along each line unless the plate/shell thickness varies and automatic calculation of eccentricity is requested.



section through shell and beams
with no eccentricities

section through shell and beams
after CALCULATED-NEGATIVE-Z-OFFSET

Figure 3.49 Automatically calculated eccentricity for L- and I-sections

3.5.4 Thickness

All surfaces for which 2-D elements (membrane, plate and shell elements) are created must be assigned a thickness. (Surfaces with axi-symmetric elements, however, need no thickness.) The PROPERTY THICKNESS command is used for this purpose. Observe that rather than giving a single value for the thickness, i.e. a constant value over the surface, a varying thickness can be defined by varying value input; see Section 3.7.

3.5.5 Local Coordinate System for Surface Elements

For most 2-D elements (membrane, plate and shell elements) the local coordinate system is not defined within Prefem. The local z-axis (surface normal) is implicitly defined by the way the surface was defined: the right-hand-rule combined with the sequence in which the lines where referred to when defining the surface determines the general direction of the local z-axis. The exact direction of the local z-axis and also the orientation of the local x-axis and y-axis are determined by the analysis program, e.g. Sestra, depending on the type of element.

The direction of the z-axis may, however, be reversed by the CHANGE ROTATION-OF-SURFACE and CHANGE NORMAL-OF-SURFACE commands. Also see Section 3.12.3.

The layered element type, however, require the local coordinate system to be defined, i.e. the orientation of the local x- and y-axes. This is done by the PROPERTY LOCAL-COORDINATE-SURFACE command. Note that this command will have no effect for other than layered elements.

3.5.6 Material

The PROPERTY MATERIAL command is used for defining the material types:

- Elastic – for all element types except for sandwich elements

- Anisotropic – for 2-D and 3-D elements for which orthotropic or anisotropic material is desired, also for sandwich elements (option 3D-SHELL-ELEMENT) for giving different (anisotropic) material for the layers

- Spring – for axial and to-ground spring elements, note that the spring material must be defined prior to defining the spring elements (see the DEFINE SPRING command), this is in contrast with what is the case for other elements and their material

- Damper – for axial and to-ground damper elements, note that the damper material must be defined prior to defining the damper elements (see the DEFINE DAMPER command), this is in contrast with what is the case for other elements and their material

- Mass – for one node mass elements, note that the one node mass material must be defined prior to defining the mass elements (see the DEFINE MASS-ELEMENT command), this is in contrast with what is the case for other elements and their material

The various materials defined are given unique names and the materials are then assigned to the appropriate geometry for which elements are created. This assignment is performed by the CONNECT MATERIAL command by referring to the material names and geometry names. Assigning materials to geometry for which no elements are created has no consequence.

3.5.7 Boundary Condition

The PROPERTY BOUNDARY command is used for defining the boundary conditions. Note that individual degrees of freedom (d.o.f.s) of the nodes may be given different types of boundary condition. Also note that a boundary condition is always given for all six d.o.f. of the nodes even in cases where only three d.o.f. exist (membrane and solid elements), the superfluous boundary conditions are simply neglected. The boundary conditions are:

- Free — the d.o.f. is not fixed or given any other special boundary condition, all d.o.f. will be free for nodes not given any boundary condition

- Fix — the d.o.f. is fixed at zero displacement

- Prescribed displacement — the d.o.f. is given a prescribed displacement, the value of the displacement is given in the PROPERTY LOAD command

- Prescribed acceleration — the d.o.f. is given a prescribed acceleration, the value of the acceleration is given in the PROPERTY LOAD command

- Super — the d.o.f. is a super d.o.f.

Boundary conditions may be given for any geometrical entity, i.e. bodies, surfaces, lines and points. A boundary condition given for a body will also apply to the surfaces, lines and points of which the body is composed. The same is the case for a surface and a line: the boundary condition given will always apply to the lower level geometrical entities of which the specified geometry is composed.

A specification of boundary condition will replace any previous specification for the same geometry. This allows a point and a line connected to it easily to be given different boundary conditions: Simply give the boundary condition for the line first and thereafter for the point. Giving the boundary condition for the line last will overrule any previously given boundary condition for the point.

Boundary conditions may be specified in the cartesian coordinate system of the model, in a rotated cartesian coordinate system or in cylindrical and spherical coordinate systems.

3.5.8 Linear Dependency

Linear dependency is forcing the displacement of a degree of freedom (d.o.f.) of a node to be a linear function of the displacement of one or several other d.o.f. The PROPERTY LINEAR-DEPENDENCY command is used for this purpose. There are a few alternative ways of defining linear dependencies:

- TWO-POINT-DEPENDENCY — defines linear dependency between one point (or rather the node in that point) and two other points (nodes in those points). The linear dependency will be valid for all d.o.f.s of the nodes.

- TWO-NODE-DEPENDENCY — defines linear dependency between one node and two other nodes. The difference between this and the previous item is that TWO-POINT-DEPENDENCY refers to nodes through point names while TWO-NODE-DEPENDENCY refers to node numbers directly and will be deleted if the mesh is deleted.

- GENERAL-NODE-DEPENDENCY — defines linear dependencies between any d.o.f. of a node and any other d.o.f.s of any other nodes. As the command refers to node numbers directly the property will be lost if the mesh is deleted.

- LINE-LINE-DEPENDENCY — defines linear dependencies between all or selected d.o.f.s of nodes on a pair of lines. The pair of lines must have exactly the same location (overlap). This feature is convenient for coupling adjoining surfaces that have overlapping lines rather than sharing common lines. Such overlapping lines for adjoining surfaces is in conflict with normal modelling practice but may be used in certain situations, e.g. to let the adjoining surfaces have different number of elements in the same position or incompatible element types (4 and 8 node shell elements).

- RIGID-BODY-DEPENDENCY — defines linear dependencies between all or selected d.o.f.s of several nodes and a single node. This feature is convenient for making an infinitely stiff coupling between a selected geometry and a point, e.g. for coupling the end of a tube modelled by shell elements to a beam element.

The boundary condition ‘linear’ is automatically introduced for the dependent d.o.f. when using either of the alternatives above. When displaying the mesh and labelling node symbols these nodes will appear as blue triangles.

Note: If the dependent node (or d.o.f.) already has a boundary condition, e.g. fixed or super, then it is changed to ‘linearl’. However, if the current boundary condition is prescribed or ‘superl’ (see below for this boundary condition code) then the boundary condition is not changed rather a warning is given.

For the independent or governing d.o.f. there are two alternatives:

- The independent d.o.f. may automatically be given the boundary condition ‘superl’. This is the default condition but if you want to set this mode then give the command SET DEFAULT LINEAR-DEPENDENCY-MODE FORCE-TO-SUPER *prior* to the PROPERTY LINEAR-DEPENDENCY command. ‘Superl’ is in effect the same as ‘super’, the ‘l’ merely indicates that the d.o.f. is super because another d.o.f. is linearly dependent of it. Such a ‘superl’ d.o.f. may also be used as a normal super d.o.f. for coupling superelements. Even if the FE model created comprises the whole structure (the structure is not

divided into several superelements) Presel must be used to assemble the complete model as a second level superelement when linear dependencies have been defined in this way. Either of Sestra's equation solvers Multifront and Supermatrix may be employed.

- The independent d.o.f. may have any boundary condition, i.e. it is not required to be 'superl' as in the other alternative. This alternative is chosen by giving the command SET DEFAULT LINEAR-DEPENDENCY-MODE NO-FORCE-TO-SUPER *prior* to the PROPERTY LINEAR-DEPENDENCY command. Also, this alternative requires use of the Multifront equation solver in Sestra.

Note: The independent node (or d.o.f.) cannot be linearly dependent of another node (or d.o.f.).

3.5.9 Point Mass

Point masses are defined by the PROPERTY POINT-MASS command. Such a mass will contribute to the mass matrix as a diagonal mass matrix as opposed to the mass element which is comprised of a full mass matrix.

3.5.10 Numeric Value Input

Numeric values need to be given for most properties. In most cases only constant values are accepted. Alternatively to giving these constant values as single real or integer values many of them can be given as *real expressions*. In such cases the program prompt will be 'real expression'. Examples of real expressions are:

- (0.3 * 1.1)
- (4 / 1.5 * 2)
- (PI ** 2)

However, in the case of surface thickness and loads (see Section 3.6) variable values may optionally be given as functions; see Section 3.7.

3.6 Defining Loads

Loads are defined by the PROPERTY LOAD command (except for initial displacements and velocities, see below). Loads are assigned to the geometry model and will automatically be transferred to the FE model, deleting and recreating the FE mesh will not affect the loads. However, loads defined for geometry where there is no mesh will be ignored. Loads are specified as constant values or as variables (by use of functions) and they may be given in the global (the superelement's) cartesian coordinate system or in user-defined cylindrical or spherical coordinate systems. The loads may also be given in transformed coordinate systems, i.e. cartesian transformations with respect to the relevant cartesian, cylindrical or spherical coordinate system.

The load types are:

- CONCENTRATED loads in points — transferred to nodes as nodal loads.
- BEAM-CONCENTRATED loads on beam elements — transferred as very short line loads.

- LINE-LOAD on lines — transferred as line distributed loads on either beam, membrane or shell elements.
- LINE-MOMENT on lines — transferred as line distributed loads on either three node beam or shell elements.
- PART-LINE on lines — transferred as line loads on parts of either beam, membrane or shell elements.
- NORMAL-PRESSURE on surfaces — transferred as normal surface pressure on shell or solid elements.
- COMPONENT-PRESSURE on surfaces — transferred as surface pressure in three components on shell or solid elements.
- TEMPERATURE loads for lines, surfaces and bodies — transferred to membrane, shell and solid elements.
- GRAVITY load — transferred to all elements and giving inertia load according to material density and volume of elements.
- RIGID-BODY-ACCELERATION and RIGID-BODY-VELOCITY load — transferred to nodes (RIGID-BODY-VELOCITY is currently not available in the analysis program Sestra).
- PRESCRIBED-DISPLACEMENT and PRESCRIBED-ACCELERATION — transferred to nodes for which boundary conditions are set accordingly.

Additionally, there are a couple of properties defined directly under the PROPERTY command, i.e. not under the PROPERTY LOAD command but which in effect are loading conditions for dynamic analysis:

- INITIAL-DISPLACEMENT — transferred to nodes as an initial condition for a dynamic analysis.
- INITIAL-VELOCITY — transferred to nodes as an initial condition for a dynamic analysis.

All elements loads are represented as nodal intensities.

Loads are assembled in load cases numbered from 1 and up. Each load case may be comprised of several of the load types above. The same load case may also contain the same load type more than once. Specifying, for example, NORMAL-PRESSURE twice for the same load case gives the same result as specifying the sum of the two pressures as a single NORMAL-PRESSURE. However, a load case can only include one gravity load.

When defining a load the part of the model subjected to the load is identified by referring to the appropriate geometry. It is therefore the extent of this geometry and *not the function* (see Section 3.7) that limits the application of the load. This is illustrated in Example 3.4.

Example 3.4 Defining a Constant Pressure Load

A constant pressure on the surface AU11 is defined by:

```
% Define load 1:
PROPERTY LOAD 1
% Load type:
  NORMAL-PRESSURE
% Reference to geometry:
```

```
  AU11
% Load value (END means no complex load):
  200 END
% Where to apply the load:
  MIDDLE-SURFACE
```

The load value 200 is in essence a constant function unlimited in space.

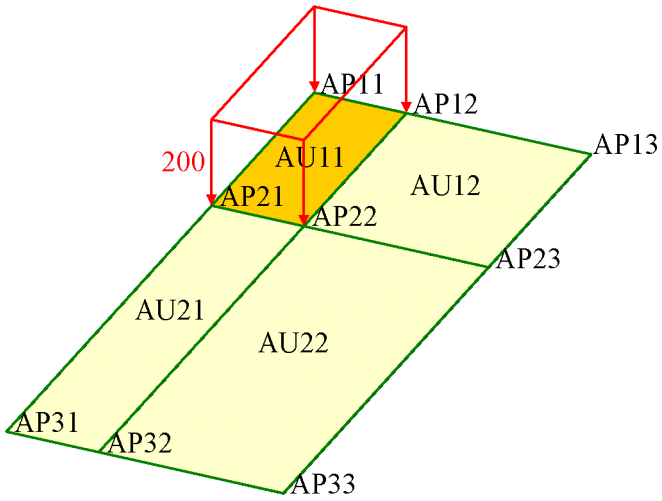


Figure 3.50 Extent of load is limited by reference to geometry

For the LINE-LOAD, PART-LINE, NORMAL-PRESSURE and COMPONENT-PRESSURE load types the concept of inside and outside of surfaces is of relevance; see Section 3.12.3.

Loads can be changed by the CHANGE PROPERTY LOAD command. (PROPERTY CHANGE is an equivalent command.) The change is performed only for the geometry that actually has loads of the given type in the given load case. The change concerns the first specified load on each geometry of correct type.

DELETE PROPERTY LOAD removes the load definition from the model. All loads in the given load case of the given load type and for the given geometry are deleted.

If changes are made to the loads by the CHANGE PROPERTY LOAD or DELETE PROPERTY LOAD commands, or the position of nodes are changed by the CHANGE NODE command, the command RE-COMPUTE LOADS will re-compute the transfer of the loads from geometry to nodes and elements. This is required to get a correct display of a load (the ADD-DISPLAY LOAD command) after a change has been made. The loads will always be re-computed when the Input Interface File is produced.

When the element mesh is changed it is best to RE-COMPUTE load cases with varying loads.

The PRINT LOAD command tabulates loads selected for load cases, load types and geometries.

The ADD-DISPLAY LOAD command displays graphically selected loads on the FE mesh currently displayed.

3.7 Varying Value Input by Functions

Surface thickness and loads (surface pressure, line loads, temperature, concentrated loads, prescribed displacements and prescribed accelerations) can be specified as values varying in space. Rather than giving a single numeric value where a value is requested a function describing the load (or thickness) variation in space may be entered. A function is defined based on a set of pre-defined basic functions, mathematical functions, arithmetric operators and constants. The various basic and mathematical functions are described in detail in Section 5.3. Introductory explanations are found below.

The basic functions available are:

- LINEAR-2POINTS-VARYING
- LINEAR-3POINTS-VARYING
- LINEAR-RADIUS-VARYING
- CYLINDRICAL-ANGLE-VARYING
- CYLINDRICAL-RADIUS-VARYING

The function may be restricted to only having a value between two points, i.e. zero value outside:

- VALUE-BETWEEN

The validity of the function may be restricted to only elements between two points, i.e. *undefined* (not the same as zero) outside:

- ONLY-BETWEEN

In addition, the following mathematical functions are available:

- SIN
- COSIN
- ABS
- SIGN
- EXP
- LN
- DIM
- SQRT
- MAX
- MIN

The constant π is entered by the parameter:

- PI

Parentheses may be used for building up expressions; remember that these are commands in themselves and must be separated from other commands and data by blanks:

- (
-)

The basic and mathematical functions above have parameters. Rather than giving numeric values for these parameters another function may be given. Furthermore, arithmetic operators may be used to add, subtract, multiply, etc. functions. In this way functions of practically unlimited complexity may be specified. The arithmetic operators are:

- + (add)
- - (subtract)
- * (multiply)
- / (divide)
- ** (exponent)

Normal operator precedence is applied, i.e. ** is calculated first, thereafter * and / while + and - are calculated last.

Example 3.5 through Example 3.8 show examples of varying value input for loads. Example 3.6 shows how a sinus function is defined by the construction:

```
SIN LINEAR-2POINTS-VARYING point1 value1 point2 value2
```

Example 3.7 illustrates how the geometry used for defining the function (points PA and PB) does not need to have any relation to the geometry subjected to the load.

See Section 5.3 for more examples of functions.

Example 3.5 Defining a Linearly Varying Pressure Load

A linearly varying normal pressure is defined as follows:

```
PROPERTY LOAD 2
  NORMAL-PRESSURE
  ( ALL-SURFACES-INCLUDED EXCLUDE AU11 )
  LINEAR-2POINTS-VARYING AP13 100 AP33 300
  END
  MIDDLE-SURFACE
```

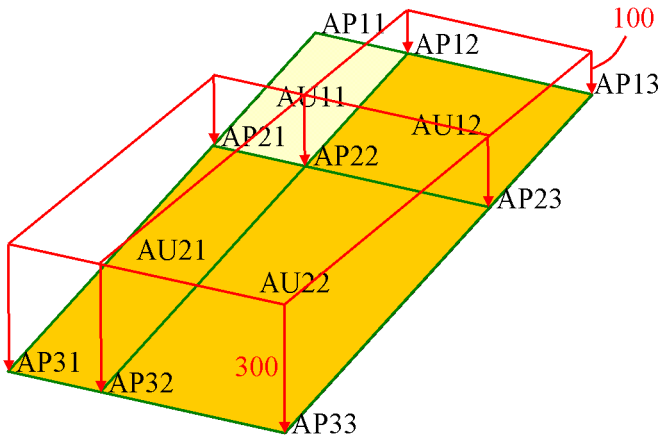


Figure 3.51 Linearly varying pressure load

Example 3.6 Defining a Sinus Varying Pressure Load

A sinus varying pressure is defined as follows:

```
PROPERTY LOAD 3
  NORMAL-PRESSURE
  ( AU12 AU22 )
  ( 200 * SIN LINEAR-2POINTS-VARYING AP11 0 AP31 PI )
  END
  MIDDLE-SURFACE
```

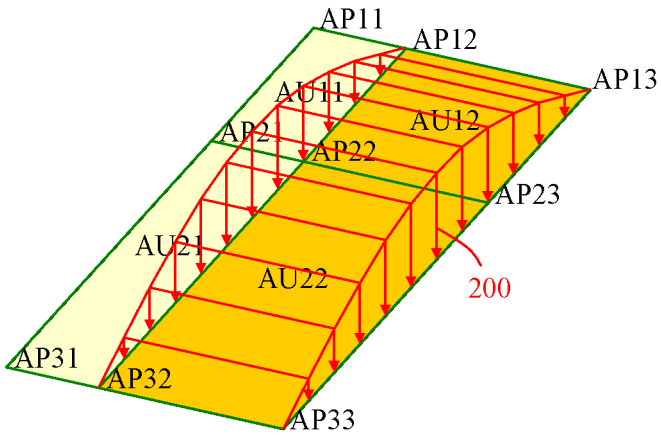


Figure 3.52 Sinus varying pressure load

Example 3.7 Defining a Hydrostatic Pressure Load

A hydrostatic pressure load is defined as follows:

```
PROPERTY LOAD 5
  NORMAL-PRESSURE
  S
  LINEAR-2POINTS-VARYING PA 0. PB 80.
  END
  OUTSIDE-SURFACE
```

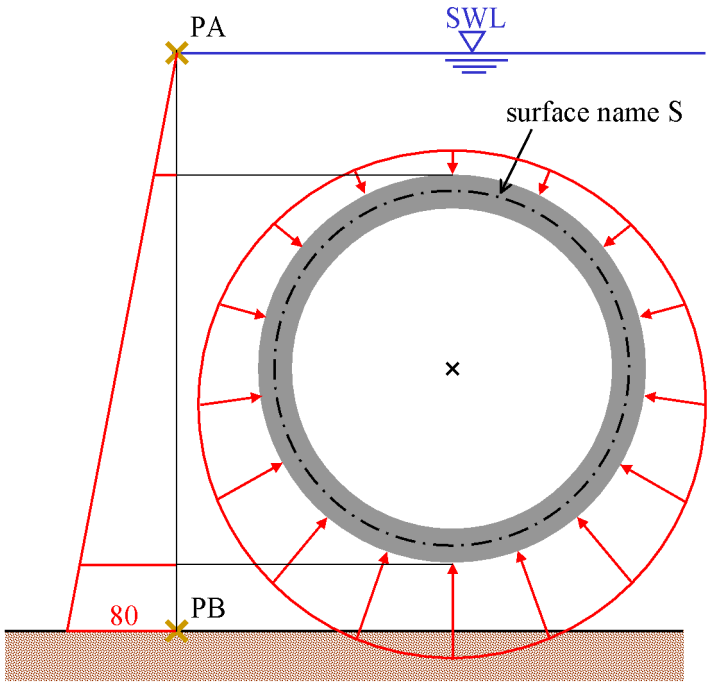


Figure 3.53 Hydrostatic pressure on sphere defined as a vertical linear variation

Example 3.8 Defining a Load Varying Linearly in one Direction and Parabolic in the Other

A linear variation in one direction and parabolic in the perpendicular direction is described by multiplying the two functions as follows:

```
PROPERTY LOAD 6
  <load type>
  <reference to geometry, e.g. surface names>
%   <----- a linear squared = parabolic function ---->
  ( ( a * ( LINEAR-2POINTS-VARYING P1 0. P2 1. ) ** 2 ) *
%   <-------- a linear function ------->
    LINEAR-2POINTS-VARYING P1 0. P3 1. )
  END
  <where to apply the load>
```

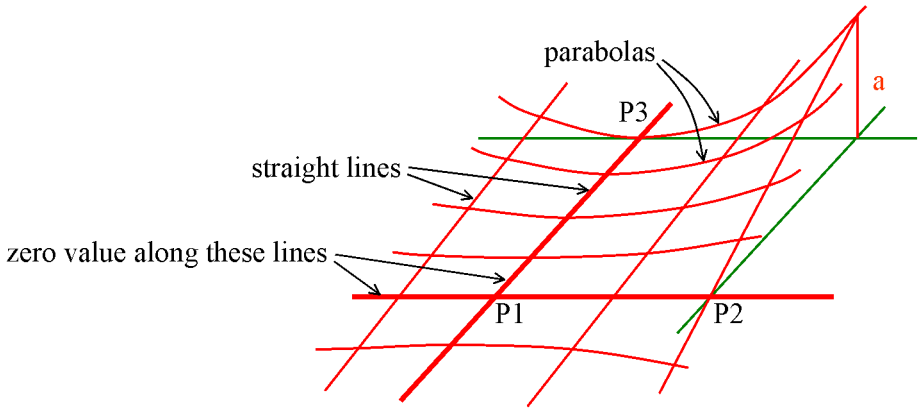


Figure 3.54 2-D function

3.7.1 Evaluation of Functions

A function is in itself unlimited in space. Its value in any given point in space (in a node in the FE model) is found by projecting the point into the function and evaluating the function there. How to project a point into a function depends on the function, the LINEAR-2POINTS-VARYING function is discussed below for illustration purposes. Full explanations are found in Section 5.3.

The LINEAR-2POINTS-VARYING function is defined by specifying two geometry points. The line between these two points is the 'projection line'. Any point on the projection line and its extension will have a value as defined by the function. This value will also apply to all points in the plane perpendicular to the projection line and through the point in question. See Figure 3.55 for an illustration of this.

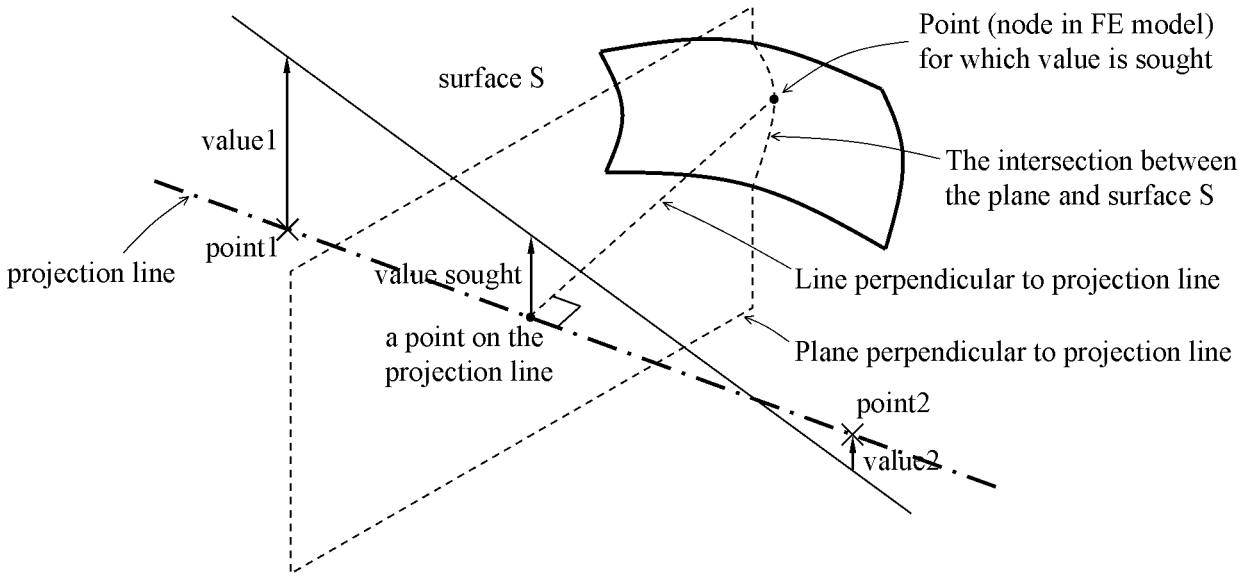


Figure 3.55 Evaluation of LINEAR-2POINTS-VARYING: point projected onto projection line

3.8 Parameters

Parameters may be defined and referred to in certain commands. This involves defining a parameter by name and giving it a value and thereafter referring to this parameter rather than specifying the value in subsequent commands. This feature opens for some degree of parametric modelling. An example is provided below.

```
DEFINE PARAMETER PAR1 value1
                 PAR2 value2
                 PAR3 value3
..
DEFINE POINT P1 < X PAR1 Y PAR2 Z PAR3 >
..
CHANGE POINT P2 < X PAR1 DY PAR2 >
..
```

Note: The command DEFINE POINT P1 PAR1 is not allowed. This because the parameter may only be used in situations where a real value is the only alternative available (and not for example END).

3.9 Selecting Geometry, using Wild-Cards and defining Sets

The following ways of selecting geometry are explained:

- Graphical selection
- Line-mode command selection
- Wild-card selection
- Selection through set

3.9.1 Graphical Selection of Geometry

Whenever there is a need for selecting geometry, e.g. when boundary conditions are defined and when loads are defined, you may select geometry graphically. There are three ways of doing this:

- Clicking the left mouse button (LMB)
- Dragging a rubber-band rectangle using the LMB
- Polygon selection: Position the cursor and press the shift key to define the first polygon point. While keeping the shift key pressed, repeatedly move the cursor and click the LMB to make a polygon. Release the shift key and click to define the last polygon point. A straight line between the first and last polygon points closes the polygon. If the LMB is pressed rather than clicked, a rubberband line appears as an aid to determine the position of the polygon segment.

Apply the above graphical selection methods as follows:

- Select a point and line/curve by the item itself or its label.

- Select a surface by rubberband/polygon around it or by clicking its label. You may also use the right mouse button (RMB) and left mouse button (LMB) in combination as follows: Click the RMB once on a borderline/curve of the surface and see that the line/curve is highlighted (changing colour). Without moving the mouse click the RMB once more and an adjoining surface is highlighted. If this is the desired surface then click the LMB. If not, then keep clicking the RMB until the desired surface is highlighted whereupon the LMB is clicked. Note that all clicks with the RMB and LMB should be done *without moving the mouse* (or at least not more than a set fractional distance).

- Select a body in the same way as explained above for a surface. When the RMB is repeatedly clicked, and after highlighting all adjoining surfaces, the adjoining bodies will be highlighted one after another.

- Note that you cannot 'loop' through the geometrical entities more than once, i.e. after highlighting the last body clicking the RMB will no longer have any effect.

The availability of graphical selection is subject to that geometry selection has been switched on by the Direct access buttons Point, Line, Surface and Body. By default they are all switched on (depressed). See information on these buttons in Section 3.1.

Note that if the Direct access button Info is depressed then geometry *cannot* be selected by clicking. See information on the Info button in Section 3.1.

3.9.2 Line-Mode Command Selection of Geometry

Geometry can be selected by line-mode commands as follows:

- Giving the geometry name directly

- Giving several geometry names within parentheses; remember to insert a space on both sides of the parentheses (the parentheses are commands in themselves):

  ```
  ( geo-name1 geo-name2 geo-name3 )
  ```

- Giving the commands ALL-BODIES-INCLUDED, ALL-SURFACES-INCLUDED, ALL-LINES-INCLUDED, ALL-POINTS-INCLUDED

- Giving wild-card type geometry names; see Section 3.9.3

- Giving name of a defined set; see Section 3.9.4

- Using the command GEOMETRY-OF-ELEMENT and giving an element number thereby selecting the geometry to which the element belongs (the geometry name is logged on the command log file)

- Excluding geometry names from geometry already selected; the following selection will comprise all lines except lines L1 and L2 (the parentheses are required or else the selection is complete after ALL-LINES-INCLUDED):

  ```
  ( ALL-LINES-INCLUDED EXCLUDE L1 L2 )
  ```

  Note that only bodies may be excluded from a selection comprised of bodies, only surfaces may be excluded from a selection comprised of surfaces, etc. The following exclusion will consequently not work (P1 is a point):

  ```
  ( ALL-LINES-INCLUDED EXCLUDE P1 )
  ```

- Including geometry names; this is relevant after an EXCLUDE command in order to counteract the exclusion. The following selection will comprise all lines except those beginning with L (see Section 3.9.3) but still including L1:

  ```
  ( ALL-LINES-INCLUDED EXCLUDE L* INCLUDE L1 )
  ```

  EXCLUDE and INCLUDE may alternate as many times as required.

Note that only the relevant alternatives for selections will be available. E.g. when defining a line load the alternatives ALL-POINTS-INCLUDED, ALL-SURFACES-INCLUDED, etc. will not be available.

The lines surrounding a surface can be selected by giving the name of the surface, lines surrounding a body can be selected by giving the name of the body, etc.

Also see Section 5.1.

3.9.3 Using Wild-Cards for Selecting Geometry

Wild-card selection and wild-card naming combined with a consistent system for naming geometry, e.g. names as produced by the GENERATE command, provides for a powerful way of referring to many geometry names.

A wild-card name is a string with the character '&' in addition to letters and digits. A wild-card name matches a name when only the character '&' is a mismatch with the name.

Trailing '&'s (e.g. AP&&&) can be replaced by a '*' (e.g. AP*) which matches any number of characters at the end of a name. '*' alone, therefore, matches all names. (It is usually more efficient, though, to use the alternatives ALL-POINTS-INCLUDED, etc. rather than '*'.)

Even when not using the GENERATE command the user may find it useful to adopt a naming scheme similar to that of GENERATE, see Section 3.3.7, as this will both enable use of the wild-card concept and make it easier to identify the different parts of the geometry model.

Whenever geometry is to be selected it is possible to use wild-card for the selection. Wild-card may in certain cases also be used when defining geometry. Wild-card used in connection with both selection and definition of geometry is exemplified by the following COPY command:

```
COPY A&1&&2 B* TRANS
```

This command will copy all geometry names starting with A and having 1 and 2 in the specified positions. These geometries, and those needed for the definition of these geometries, are all copied. The resulting geometries will be named B&1&&2.

A non-systematic naming of geometry may result in naming conflicts. For example, if a line included in the selection A&1&&2 has been defined based on point BP1342 then the name of a copy of point AP1342 (if existing) will be in conflict with the existing point BP1342. Prefem will in such cases select another name for the copy of AP1342. The copying will succeed at the cost of a system in geometry naming.

Here are two examples of wild-card used in defining lines:

```
DEFINE LINE AL&11 AP&11 AP&12
DEFINE LINE AL&11 AP&11 AP&1+
```

These commands define new lines whenever a pair of existing points matches the pair of point specifications given. The matching pair of points must have the same character in the position of the &. This character is used in constructing the name of the created line (i.e. & replaces the same character in all three names). The + and - signs may also be used implying an incrementation or decrementation of the previous point name. The + and - symbols are only allowable in the DEFINE LINE command. The commands above are functionally identical.

3.9.4 Defining and using Sets for Selection of Geometry

Named sets consisting of parts of the geometry model (bodies, surfaces, etc.) are defined by the DEFINE SET command. Such sets may be referred to when assigning properties and loads and in the DISPLAY and PRINT commands. Within the DEFINE SET command operations are performed to specify the contents of the set. The following operations are available:

- UNION-WITH
- SUBTRACT-BY
- INTERSECTION-WITH

A sequence of operations may be performed within the DEFINE SET command and once a set has been defined it may be changed by the CHANGE SET command in which the same operations are available. Within each operation bodies, surfaces, lines, etc. are referred to by their explicit names or by wild-card names (see Section 3.9.3). It is important to keep in mind that the operations are performed in the order given and that one operation may undo a previous operation.

The use of wild-cards combined with defining sets is a powerful feature that can be used both for defining the model and verifying it. The user is advised to study this feature.

Sets consisting of elements and nodes may also be defined. Such sets will, however, rely on the result of the automatic meshing. A minor change in the mesh may result in new node and element numbers and the contents of the sets will change. Therefore, defining sets consisting of geometry is generally the best approach.

An example:

```
DEFINE SET SET1
% ---------- The set is yet empty
UNION-WITH
% ---------- Geometry is now being included in the set:
SURFACES ( S1 S2 S3 )
BODIES ( B1 B2 )
END
% ---------- The set now contains 3 surfaces and 2 bodies
END
% ---------- Now, display the lines of the bodies and surfaces in SET1:
DISPLAY LINES SET1
```

3.10 Defining ‘Special’ Element Types

Most elements are created simply by setting appropriate type of element for relevant geometry (SET ELEMENT-TYPE) and then creating the FE mesh (MESH ALL); see Section 3.4.1. Properties like material,

thickness, etc. may then be defined. For certain elements the approach is somewhat different in that the appropriate material must have been defined before the elements can be created. These elements are the spring, damper and mass elements plus the layered and sandwich elements (multilayered elements). See Section 3.10.1 and Section 3.10.2, respectively.

Furthermore, certain rules must be obeyed when creating axi-symmetric elements; see Section 3.10.3.

3.10.1 Spring, Damper and Mass Elements

Spring, damper and mass elements are defined by the following sequence of commands:

- PROPERTY MATERIAL mat-name

  After giving the material name the material kind is selected. Rather than selecting ELASTIC (which is normally used) the following options are available:

  — SPRING

  — DAMPER

  — MASS

  For the SPRING and DAMPER alternatives the options AXIAL and TO-GROUND are available; for the MASS alternative the option ONE-NODED is available. Thereafter data defining the ‘material’ is given.

- Then the elements are defined by the DEFINE command in which the appropriate of the following alternatives should be chosen:

  — SPRING

  — DAMPER

  — MASS-ELEMENT

Note that the SET ELEMENT-TYPE command is not used.

3.10.2 Sandwich Elements and Layered Elements

Sandwich Elements

Sandwich elements are multilayered shell elements comprised of normally three but in principal any number of layers through the shell thickness. Sandwich elements are defined in the same way as other shell elements. Observe, however, that the number of layers and the thickness of each layer (in percentage of total shell thickness defined by the PROPERTY THICKNESS command) are defined within the PROPERTY MATERIAL command as follows:

- PROPERTY MATERIAL material-name ANISOTROPIC 3D-SHELL-ELEMENT q1 q2 q3 rho nlay

  Here nlay is the number of layers (q1, q2, q3 and rho are explained in the command description in Chapter 5).

Layered Elements

Layered elements are typically used for modelling stiffened plates. This type of element allows the stiffeners to be modelled as an integral part the plate, i.e. a layered element spans several stiffeners of the plate. See

Figure 3.56. This feature provides a solution to the dilemma of choosing between modelling all stiffeners as beams (which involves small plate elements in between all stiffeners which in turn easily yields an excessively large model) and lumping stiffeners by modelling only some of them and adding their stiffnesses (which will produce incorrect local results).

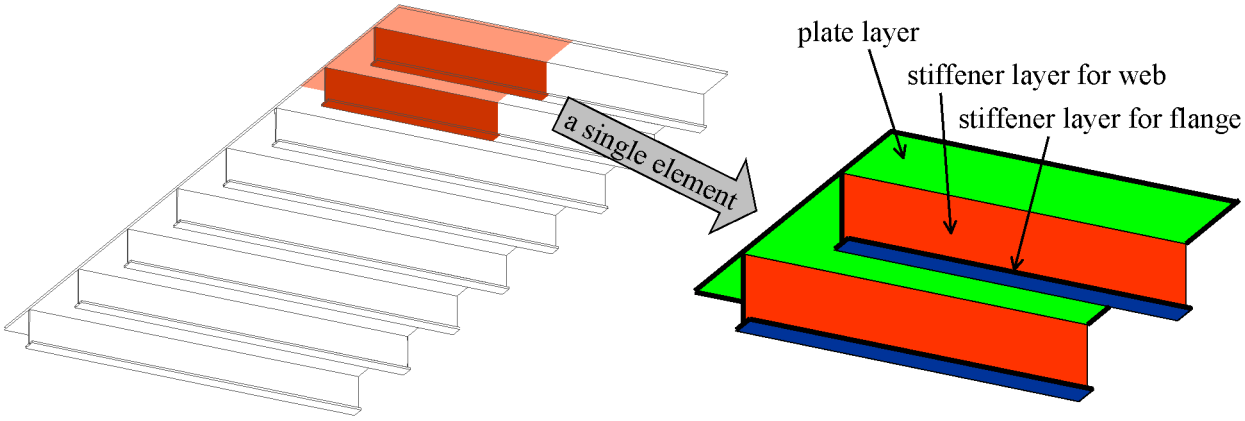
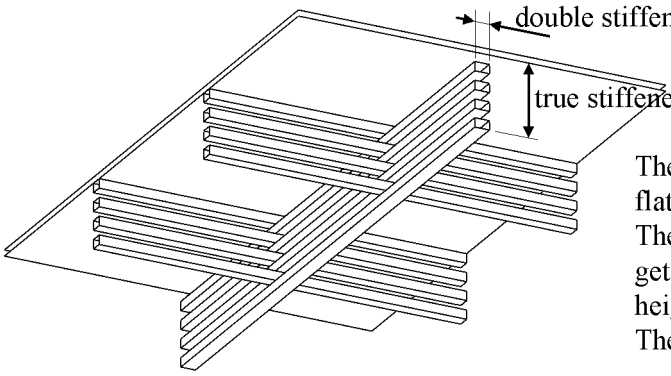

a single element
plate layer
stiffener layer for web
stiffener layer for flange
stiffened plate seen from underneath
layered element with one plate and two stiffener layers

Figure 3.56 Layered element for modelling plate and stiffener

Layered elements are multilayered shell elements comprised of any number of layers through the shell thickness. Each layer is either a plate with isotropic material properties or a stiffener layer. A stiffener layer consists of uniformly distributed stiffeners with bar sections. The material of the stiffeners is isotropic but the stiffener layer will in effect be an orthotropic layer. The layers are not allowed to overlap. (Overlapping layers will produce correct displacements but incorrect stresses.)

The restriction that layers cannot overlap implies that a plate with an orthogonal set of stiffeners cannot be modelled, i.e. a layered element can only model stiffeners in one direction. There are two solutions to this problem:

- Model the stiffeners in one direction (the stiffeners with the largest spacing, the primary stiffeners) by beam elements and include only the secondary stiffeners in the layered element.
- Include both stiffener directions in the layered element by modelling each of them by interwoven non-overlapping stiffener segments (‘fingers’) as shown in Figure 3.57. The segments must have double thickness compared to the true stiffener in order to get proper area.



The stiffeners in both directions (in this case only flatbars) have been split into interwoven 'fingers'. The thickness of the 'fingers' have been doubled to get correct areas of the stiffeners. The stiffener heights are represented by the sum of the 'fingers'. The more 'fingers' the more correct representation.

Figure 3.57 Layered element with interwoven 'fingers' for plate with orthogonal stiffeners

Layered elements are defined as follows:

- Define the material of both plate and stiffeners by the PROPERTY MATERIAL material-name ELASTIC command.
- Define the bar sections of the stiffeners by the PROPERTY SECTION section-name BAR command.
- Define the layers:

  DEFINE LAYERED layered-name PLATE ... plate-data ...

  STIFFENER ... stiffener-data ...

  The PLATE and STIFFENER alternatives are repeated to construct the complete layered element. This determines the number of layers that the layered element is comprised of.
- Select the appropriate type of layered element by the SET ELEMENT-TYPE command.
- Define local coordinate system by the PROPERTY LOCAL-COORDINATE-SURFACE command. If this command is omitted a default local coordinate system is employed.
- Connect the layered element to the appropriate surfaces by the CONNECT LAYERED command.
- Create the mesh by the MESH command.

3.10.3 Axi-symmetric Elements

Certain requirements must be met when modelling an axi-symmetric FE model. In Sestra a cylindrical coordinate system (r,z,θ) is used and the model consists of elements in the r-z-plane which corresponds to modelling in the x-y-plane in Prefem. The requirements are:

- The model must be defined in the x-y-plane and with positive x-values only; see Figure 3.58.
- Axi-symmetric elements must have anti-clockwise internal node numbering as shown in Figure 3.58. This is achieved by defining the surfaces by referring to the bordering lines in anti-clockwise direction.

- The radius (= x-axis in the model) must be zero at the axis of symmetry for all superelements. This implies that when positioning a superelement in Presel it is only allowed to translate it in y-direction and to mirror it about the x-z-plane.
- Only concentrated loads in points are allowed.

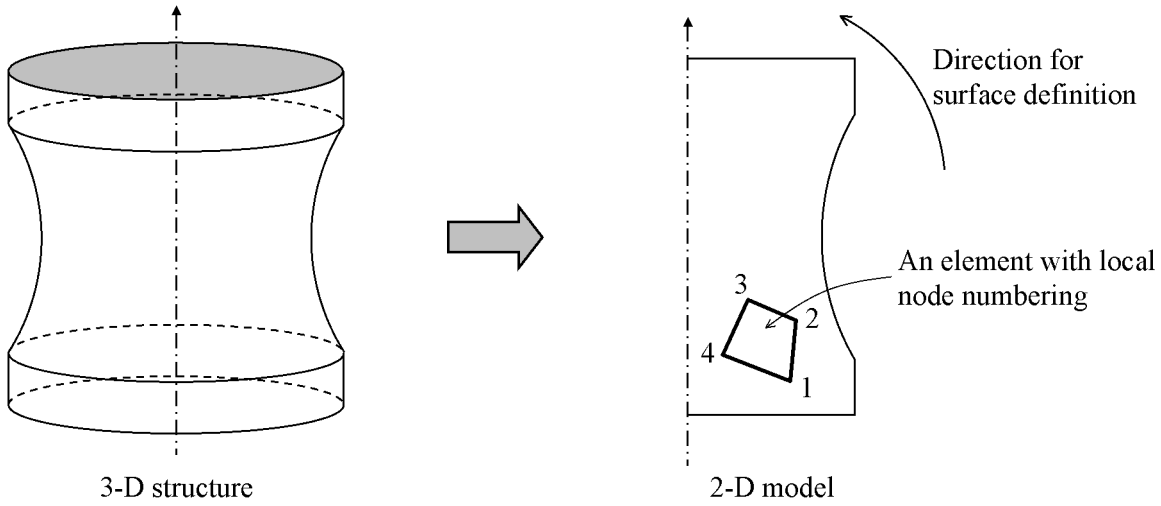


Figure 3.58 Requirements to axi-symmetric FE model

3.11 Transformations and Coordinate Systems

3.11.1 Transformations

Transformations are defined by either the DEFINE TRANSFORMATION command or the PROPERTY TRANSFORMATION command. The difference between these two commands is first of all that the former is the more powerful one: in addition to a pure rotational type of transformation, a translational, scaling and even mirroring type of transformation may be defined. Secondly, their methods for defining a rotational transformation differ. Apart from this the use of the transformations is the same. The user may find that solely using the DEFINE TRANSFORMATION command both offers him all features needed and is the most consistent approach.

A transformation is composed of any combination of:

- Translations
- Rotations
- Scaling
- Mirroring

Note: For a transformation composed of more than one of the elements above the order is generally of consequence: a translation followed by a rotation is different from the same in reversed order.

Transformations are first defined and given names. Thereafter, they are ready for use:

- For defining boundary conditions and loads in a transformed (inclined) coordinate system
- For orientating spring-to-ground and damper-to-ground elements
- For copying geometry

Note: When referring to a transformation in the definition of a load then only the rotational and mirror parts of the transformation are relevant. If the same transformation is used for defining a boundary condition and orientating spring and damper elements then only the rotational part is relevant. Translations and scaling are only relevant when the transformation is used for copying geometry.

Figure 3.20 shows an example of copying geometry by use of a transformation. Example 3.9 shows how a load may be defined in a transformed coordinate system.

Example 3.9 Defining a Load in a Transformed Coordinate System

A transformation named TR1 is defined as a 30° rotation about an axis going through the points P1 and P2. The global coordinate system (X,Y,Z) multiplied by this transformation yields the system (X_T,Y_T,Z_T); see Figure 3.59. A load may be defined in this transformed coordinate system as follows:

```
DEFINE TRANSFORMATION TR1 ROTATE ANGLES 30 P1 P2 ;
PROPERTY LOAD ... TRANSFORMED TR1 ...
```

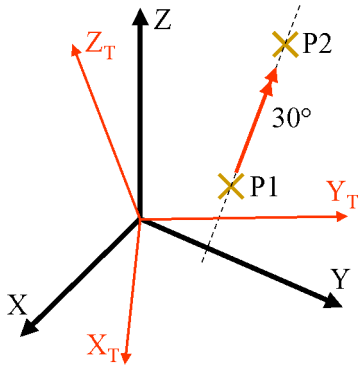


Figure 3.59 Use of transformation for defining loads

3.11.2 Coordinate Systems

The DEFINE COORDINATE-SYSTEM command defines named coordinate systems for use in other commands. Any number of cylindrical and spherical coordinate system may be defined. Relative to the cartesian coordinate system of the model the following data is given (same as within the GENERATE command):

- Type of coordinate system, i.e. CYLINDRICAL or SPHERICAL
- The origin of the cylindrical or spherical coordinate system
- Coordinates of a point determining the Z-axis (cylindrical) or pole axis (spherical)
- Coordinates of a point determining the plane where Φ=0

Coordinate systems are used:

- For defining point coordinates
- By the GENERATE command (note that the GENERATE command will, unless referring to a pre-defined coordinate system, implicitly define a coordinate system)
- For defining boundary conditions and loads. Radial and circumferential boundary conditions are for example easily defined by referring to a cylindrical or spherical coordinate system.

A coordinate system may for example be used for specification of point coordinates as shown by the example below.

```
DEFINE COORDINATE-SYSTEM CY1 CYLINDRICAL start-xyz z-axis-xyz r-axis-xyz
DEFINE POINT P1 < USE-LOCAL-COORDINATE-SYSTEM CY1 R r PHI phi Z z >
```

The parameters start-xyz, z-axis-xyz and r-axis-xyz are the three sets of coordinates defining the origin, Z- or pole axis and Φ=0 plane. Compared with the description for updating point coordinates in Section 3.3.1 the options X, Y, Z, DX, DY and DZ are replaced by R, PHI, Z, DR, DPHI and DZ for cylindrical coordinate systems. For spherical coordinate systems the options are R, PHI, THETA, DR, DPHI and DTHETA.

Loads and boundary conditions may be given in a pre-defined named coordinate system by entering the command alternative LOCAL-COORDINATE-SYSTEM followed by the name of the system. The loads and boundary conditions are then given in the R, Φ, Z system for cylindrical coordinate systems and in R, Φ, Θ for spherical systems.

Coordinate systems defined are printed on the screen by the command PRINT TRANSFORMATIONS.

3.12 Verifying and Checking the Model

The importance of checking and verifying both the geometry model and the FE model is self-evident and should be conducted according to the users best judgement. The following sections describe some useful features for this purpose. In addition to the features presented below the PRINT command, providing more exact information than the DISPLAY command, is useful for verifying and documenting the model.

3.12.1 Display and Plot Features

Figure 3.60 through Figure 3.65 show some of the features for displaying the model. The user is advised to try out various display features while referring to the command description in Chapter 5 thereby gaining sufficient skill in displaying the model for verification and reporting purposes.

Note: For large models the DISPLAY command may require some time. During processing of the command you may use Shift+Esc to abort it. This will have no effect on the program execution

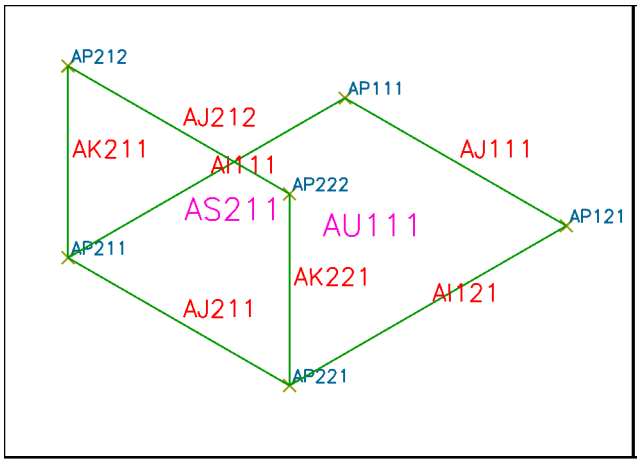
(apart from incomplete execution of the command in question) and you may continue modelling. This is available on PC only; a similar feature is not available on other operating systems.

The PLOT command produces a plot file of the current display (or sends the plot directly to the printer depending on the chosen plot format). The SET PLOT command changes the format as well as the name of the plot file to produce.

The CGM-BINARY plot format may be imported in MS Office Word and PowerPoint and even converted in these programs to an MS Office drawing thereby allowing it to be modified. (Problems importing a CGM-file may be caused by lack of a graphics filter for CGM in your installation of Word and PowerPoint. Go to a MS support or knowledge base web page to learn what to do.)

The PostScript plot format is an ASCII file and may also be imported in MS Office Word and PowerPoint as well as other word processing programs.

The geometry model in the following figures consists of two surfaces while the FE model consists of eight 4 node shell elements and six 2 node beam elements. Note that the commands given in the figures are not necessarily complete in that intermediate commands (like END or ..) may have been omitted. Some of the commands are also abbreviated.



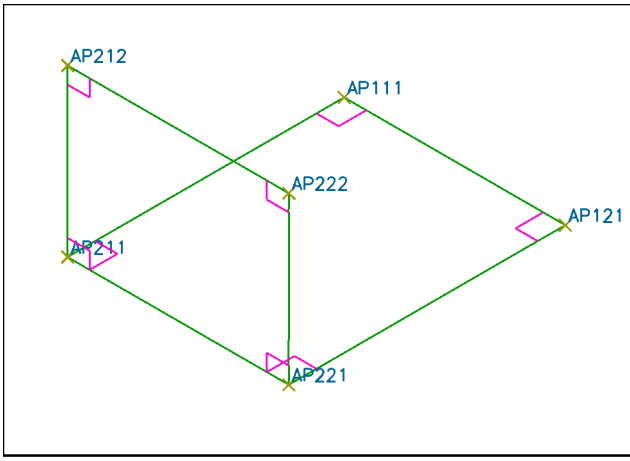
The geometry with point, line and surface names:

```
DISPLAY GEOMETRY
LABEL POINT-NAMES ON
LABEL LINE-NAMES ON
LABEL SURFACE-NAMES ON
```

Only points:

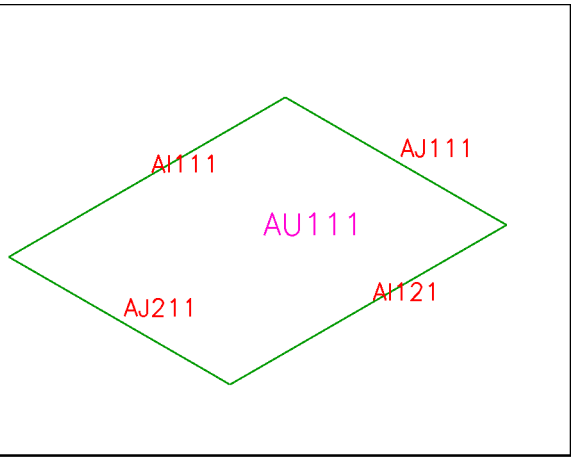
```
DISPLAY POINT ALL-POINTS-INCLUDED
```

Figure 3.60 Display features 1



The geometry with mesh corners (and point names):

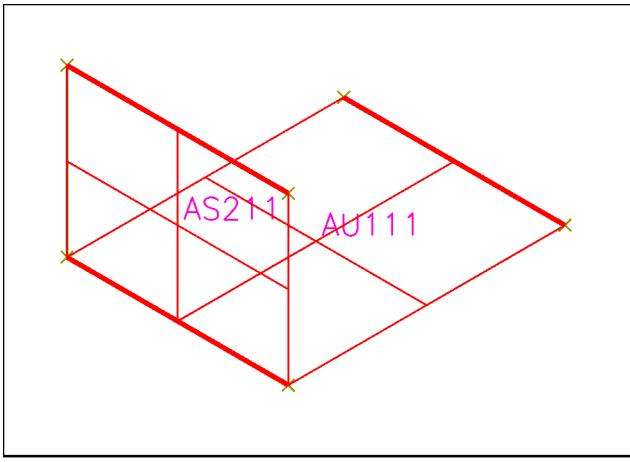
```
LABEL MESH-CORNER ON
```



Only selected surface (and line and surface names):

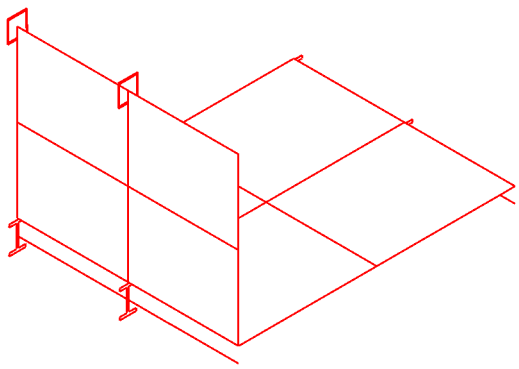
```
DISPLAY SURFACE AU111
```

Figure 3.61 Display features 2



The geometry (with surface names) with mesh added (beams drawn between nodes with thick lines):

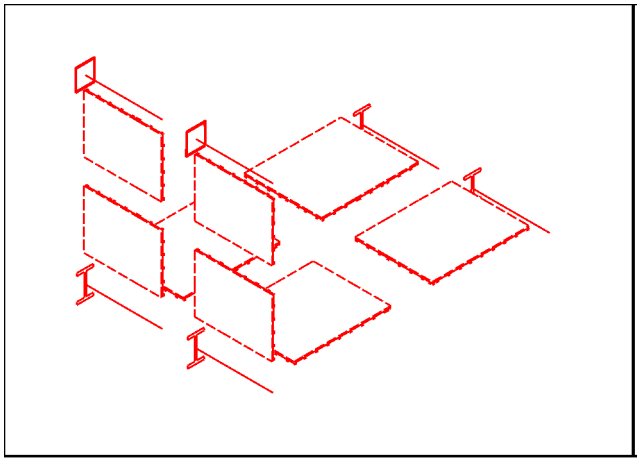
```
ADD-DISPLAY MESH
```



Only mesh, with hidden lines removed and outline of eccentric beam sections (I and box sections):

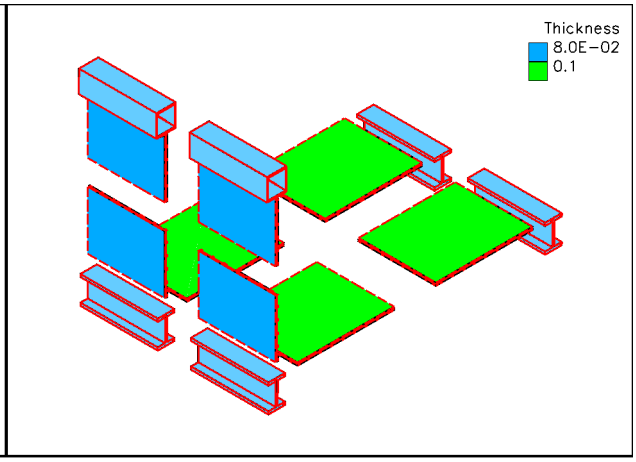
```
SET GRAPHICS HIDDEN ON
SET GRAPHICS PRESENT BEAM-ELEMENT
                     OUTLINE-SECTION
DISPLAY MESH
```

Figure 3.62 Display features 3



Shrunken elements and element thickness:

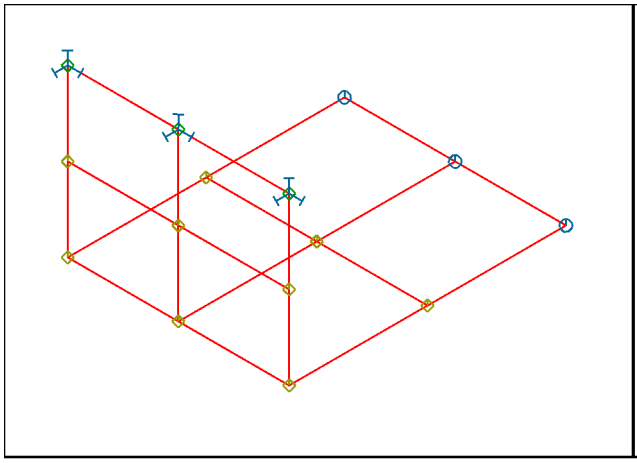
```
SET GRAPHICS SHRINK-FACTOR 0.7
SET GRAPHICS PRESENT ELEMENT-THICK
                     SYMBOLIC
LABEL ELEMENT-THICKNESS ON
```



Colour coding of thickness and beams as solids:

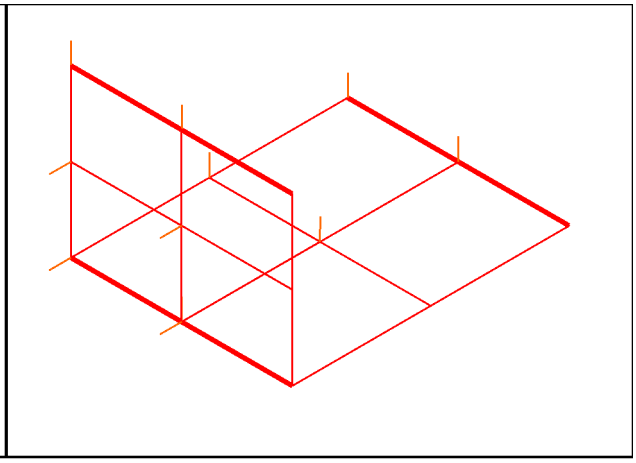
```
SET GRAPHICS PRESENT FILL-ELEM ON
LABEL COLOUR-IDENTIFIC THICKNESS
          DISCRETE-VALUES ABOVE 0 0
SET GRAPHICS PRESENT BEAM-ELEMENT
                     SECTION-AS-SOLID
```

Figure 3.63 Display features 4



Labelling of nodes (free: yellow diamonds, fixed: green diamonds, super: blue circles) and boundary conditions:

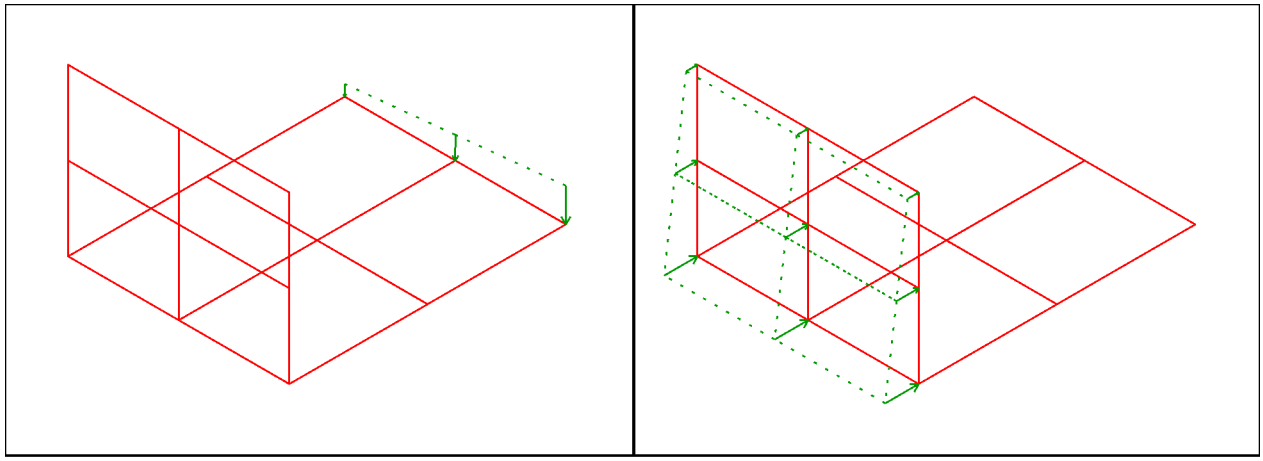
```
LABEL NODE-SYMBOL ON
LABEL BOUNDARY-CONDITION-SYMBOL ON
```



Labelling of element-normal (the in-plane axes are determined by the analysis program):

```
LABEL ELEMENT-NORMAL ON
```

Figure 3.64 Display features 5



Loads may be displayed, load case 1 is a line load:

```
ADD-DISPLAY LOAD 1 LINE-LOAD
```

Load case 2 is a normal pressure:

```
ADD-DISPLAY LOAD 2 NORMAL-PRESSURE
```

Figure 3.65 Display features 6

3.12.2 Checking the FE Mesh

The CHECK MESH-TOPOLOGY command is used to check whether it is possible to create a mesh for the whole or parts of the geometry model. A mesh is not created. The CHECK MESH-TOPOLOGY command is convenient as it is significantly quicker than the MESH command.

The quality of the mesh is investigated using the CHECK ELEMENT-SHAPE command. There are options for checking for:

- Maximum and minimum angle of elements corners
- Aspect ratio of the elements, i.e. largest distance between two corner nodes divided by shortest
- Twist of elements

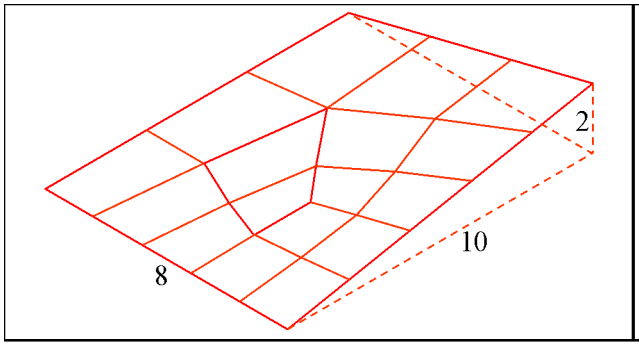
In addition to specifying the type of check to perform and limiting values the CHECK ELEMENT-SHAPE command will request the name of a *set*, see Section 3.9.4, into which the elements failing the test will be put. This set containing 'bad' elements may then be displayed. The command sequence will typically be:

```
CHECK ELEMENT-SHAPE geometry set-name1 ANGLE MAXIMUM/MINIMUM value
                            set-name2 ASPECT-RATIO value
                            set-name3 TWIST value
DISPLAY geometry
ADD-DISPLAY ELEMENT set-name1/set-name2/set-name3
```

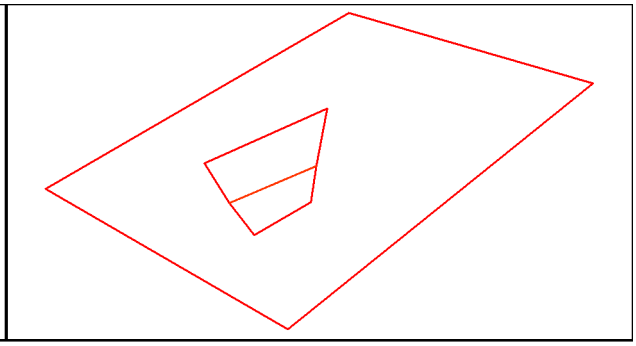
An alternative way of displaying the 'bad' elements is:

```
DISPLAY ELEMENT set-name1/set-name2/set-name3
LOCATE
```

Figure 3.66 illustrates the use of the CHECK ELEMENT-SHAPE command.

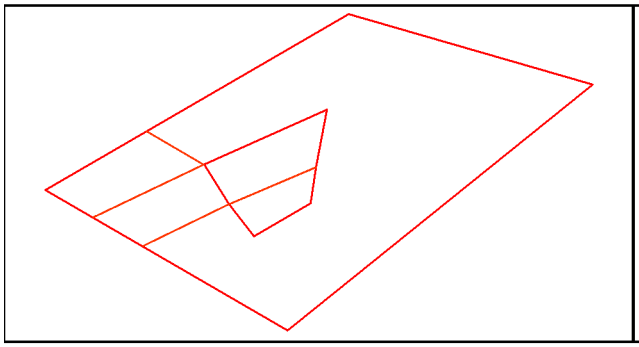


The mesh for a twisted surface. The surface with dimensions 8 by 10 has its fourth corner out of plane by a distance of 2. The 4 node elements will then be somewhat twisted.



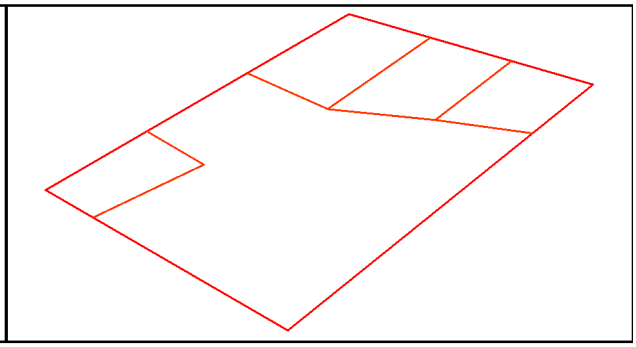
Check for angles less than 60°:

```
CHECK ELEMENT-SHAPE ALL-SURF-INCL
      SET1 ANGLE MINIMUM 60
```



Check for aspect ratio greater than 2:

```
CHECK ELEMENT-SHAPE ALL-SURF-INCL
      SET2 ASPECT-RATIO 2
```



Check for twist greater than 4°:

```
CHECK ELEMENT-SHAPE ALL-SURF-INCL
      SET3 TWIST 4
```

Figure 3.66 Checking FE mesh for angles, aspect ratios and twist

3.12.3 Modelling Considerations

Element local z-axis

For 2-D surface elements the direction of the local z-axis of the elements is decided in Prefem while the local x- and y-axes are determined by the analysis program, e.g. Sestra. (The *exact* direction of the local z-axis is also determined by the analysis program as the exact element shape is only known by the analysis program.) The local z-axis is governed by the rotational order of the lines/curves defining the surface combined with the right-hand-rule; see Figure 3.67.

Note: Coplanar neighbour surfaces defined with opposite rotational order will give neighbour shell elements with opposite local z-axes. Principally, there is nothing wrong in this but the interpretation of the results will be confusing and prone to error as corresponding stress components belong to opposite shell surfaces.

The rotational order of a surface and thereby the local z-axis of its elements is changed by the CHANGE ROTATION-OF-SURFACE command. Alternatively, the CHANGE NORMAL-OF-SURFACE command may be used.

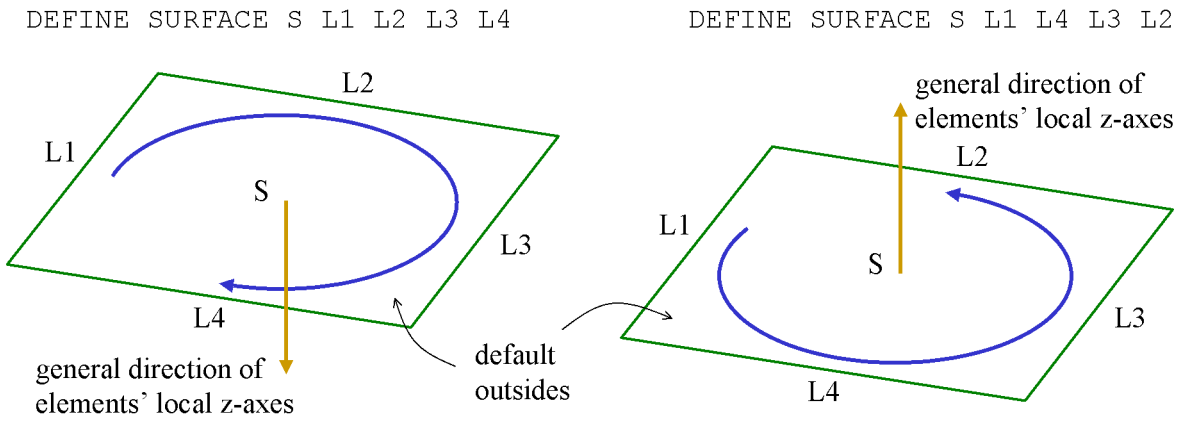


Figure 3.67 Elements' local z-axis is determined by surface definition

Inside/Outside of Surfaces

Loads are applied to the inside, middle or outside of surfaces. Reversing the inside and outside of a surface has the following effect; also see Figure 3.68:

- For LINE-LOAD, PART-LINE, COMPONENT-PRESSURE and NORMAL-PRESSURE the area (length for LINE-LOAD and PART-LINE) to which the load is applied may change and thereby the load sum will change.
- For NORMAL-PRESSURE and HYDRO-PRESSURE the direction of the load will also change.

Note: The concept of inside and outside of surfaces has relevance for the application of loads only.

By default the outside of a surface is on the positive local z-side; see Figure 3.67. This default condition is changed by the SET INSIDE/OUTSIDE commands. These commands will, however, *not* change the rotational order and thereby *not* the local z-axis of the elements.

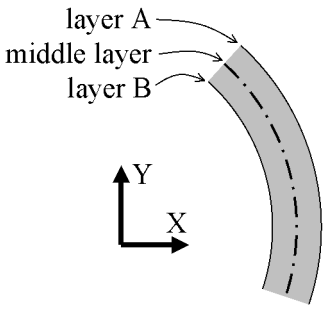
The user may find it more convenient to use either the CHANGE NORMAL-OF-SURFACE or the CHANGE ROTATION-OF-SURFACE command to maintain a consistency between inside/outside and the local z-axes.

Note: The direction of the local z-axis has no influence on the load application.

Note: A surface can only have one setting of inside/outside. You cannot have different inside/outside setting for different loads.

Note: The evaluation of functions, see Section 3.7.1, is based on nodal coordinates. This means that while the thickness of a shell will influence the application of a load as illustrated in Figure

3.68, it will not influenced the load intensities found by evaluating the specified function. For very thick shell structures this may have a significance.



The table below shows how pressure loads defined for the *outside* of a curved surface depend on which surface is the outside one. The loads are COMPONENT-PRESSURE, a *positive* NORMAL-PRESSURE and a *negative* NORMAL-PRESSURE.

The middle layer has the same position as the geometry surface.

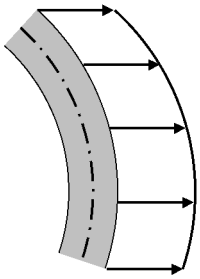
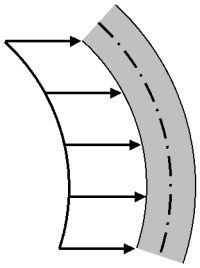
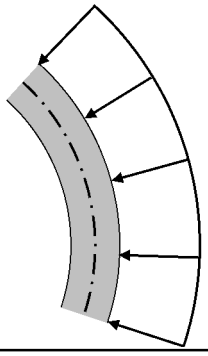
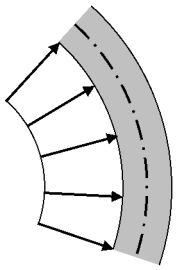
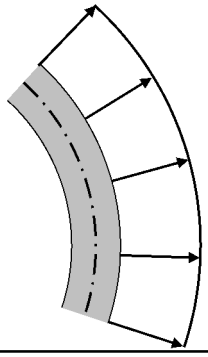
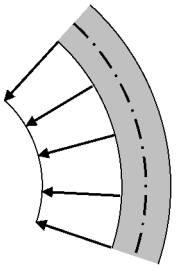
	layer A is outside load sum will be 'high'	layer B is outside load sum will be 'low'
positive X-component of COMPONENT-PRESSURE The direction of the load does not change when outside layer changes from A to B.		
positive NORMAL-PRESSURE The direction of the load changes when layer B is made outside because positive pressure points from outside towards inside.		
negative NORMAL-PRESSURE The direction of the load changes when layer B is made outside because negative pressure points from inside towards outside.		

Figure 3.68 Pressure loads applied to surface

Inside/Outside of Bodies

When defining a pressure load for a body the surfaces subjected to the pressure are referred to in the command. The inside of a body is automatically determined by the program as the side of the surface where there are solid elements. This cannot be changed by the SET INSIDE/OUTSIDE command or by any other command. See Figure 3.69.

Note: The concept of inside and outside of bodies has, as for the surfaces, relevance for the application of loads only.

Note: When there are shell elements on the surface of a body with solid elements (a type of modelling which may be employed for a sandwich type structure and when the use of sandwich and layered elements for some reason is not desired) then the definition of inside/outside will be determined by the surface and not by the body. This may involve that the direction of a NORMAL-PRESSURE load will change depending on whether a layer of shell elements are found on the surface of the solid elements or not.

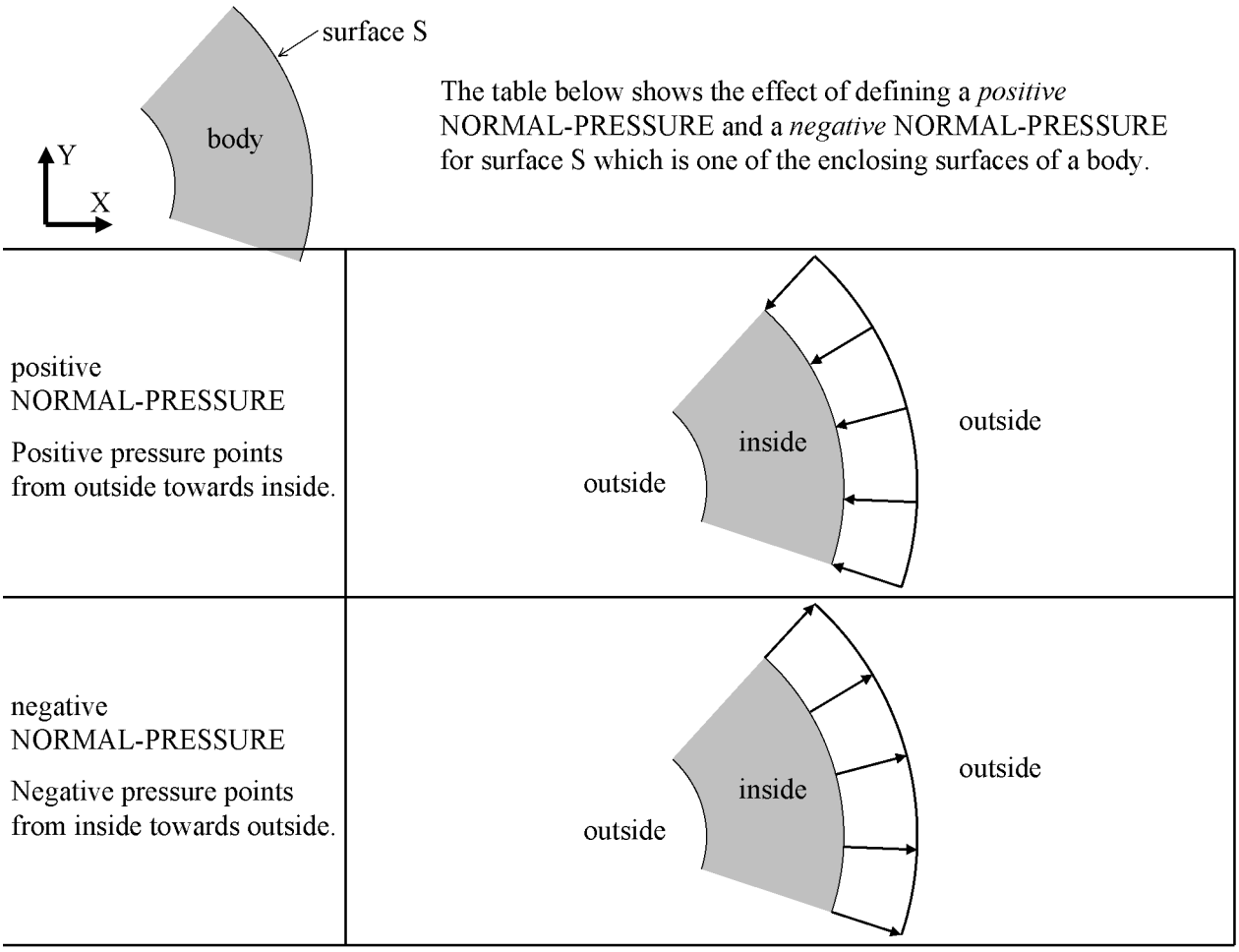


Figure 3.69 Pressure loads applied to body

4 EXECUTION OF PREFEM

This section provides information on:

- How to start Prefem
- How to read a Command input file into Prefem
- How to execute Prefem outside Manager, Unix only
- Line-mode input syntax
- Files used
- Creating plots for reports
- Alternative execution modes
- Program requirements
- Program limitations

4.1 Program Environment

Prefem is available in the following hardware environments:

- Unix computers of various vendors
- Windows 98, NT, 2000 and XP, often referred to as PC

4.1.1 Starting Prefem from Manager

Prefem is started from Manager by clicking **Model | General Prefem**; see Figure 3.3. The graphical user interface of Prefem is explained in Section 3.1.

On Unix the graphical user interface is based on OSF/Motif X Window System.

4.1.2 Reading a Command Input File into Prefem

In the General Modelling window opening up when giving Model | General Prefem in Manager there is a box for specifying a Command input file; see Figure 4.1. By default this is set to None. Changing this to File name a new box appears in which you may specify a Command input file that will be automatically read into Prefem once the program is started by clicking OK. If the box Run interactively after command input file processing is checked Prefem will display the geometry model created by the input file and await interactive user input. You may then continue modelling or only verify the current model and leave Prefem by the EXIT command.

Note: If your Command input file constitutes the complete input then make sure the Database status is set to New. You need to change Old to New if you previously have run Prefem in which case there will exist a Prefem database causing the Database status to come up as Old.

Note: On the other hand, if your Command input file shall be added to an existing model then leave Database status as Old. In this way you may repeatedly add Command input files to build up your complete model. You may for instance first read a file containing definition of your preferred beam cross section types, leave Prefem and then read the modelling input referring to these cross section types.

Note: You may also read a Command input file from inside Prefem by using the SET COMMAND-INPUT-FILE command followed by the # command; see these.

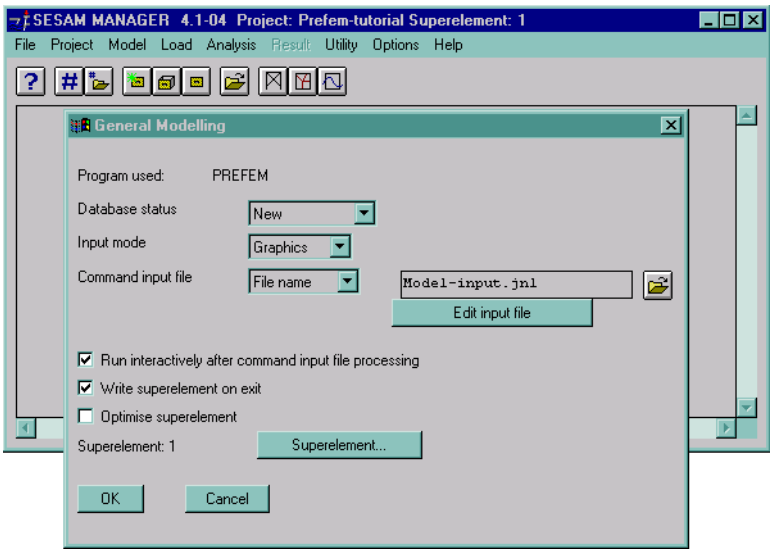


Figure 4.1 Manager and the General Modelling window with Command input file specification

4.1.3 Starting Prefem as an Individual Program on Unix

Alternatively to starting Prefem from Manager it may on Unix be started as an individual program. Provided SESAM is properly installed on your computer Prefem is started by the command ‘prefem’ (in lower case).

The program responds by presenting itself and giving some key information:

- Program version number and release date (date of executable file)
- Access date and time (now)
- Your user identification and other operating system and hardware related data

Before you are allowed to continue, i.e. start giving commands, you must respond to the following requests for information:

- ‘General file name prefix’

  A character string forming a part of (by preceding) the names of the files opened by the program. It may and may not include a directory specification.

- ‘Model file name’

  The name of the ‘model file’ (a binary data base file), the ‘command log (journal) file’ and other files opened by the program; see Section 4.1.5.

- ‘Old or new file’

  Choose ‘new’ when commencing modelling and ‘old’ when continuing an earlier session, i.e. the ‘model file’ and ‘command log file’ already exist.

4.1.4 Line-Mode Input of Commands and Arguments

Having successfully entered Prefem as explained in Section 4.1.3 the command prompt appears, the ‘#’ character. The window in which you are now working is called the line-mode window. You may switch to the graphical user interface explained in Section 3.1 by the line-mode command SET GRAPHICS INPUT ON or you may continue working in the line-mode window (displaying the model will then open a display window).

The information below is about entering line-mode commands. Note that line-mode commands may also be entered in the graphic-mode window. The information below is, with a few exceptions, therefore also relevant for the graphical user interface.

The syntax and characteristics of line-mode input are as follows:

- The parameters (commands, sub-commands and data) are separated by one or more blank characters (or a comma) and may be entered one by one or with two or more entries on a single line of input. For example:

```
COMMAND
SUB-COMMAND
SUB-SUB-COMMAND
data ...
```

is equivalent to:

```
COMMAND SUB-COMMAND SUB-SUB-COMMAND data ...
```

Note, however, that data belonging to different data sets cannot be entered on a single line.

- UPPER CASE = lower case (all commands will be logged on a ‘command log file’ in UPPER case).
- Commands and sub-commands may be abbreviated as long as they are unique. In a command consisting of words separated by hyphens, each word may be abbreviated or completely left out. Examples:

  ```
  NODE-NUMBERS = N-N
  COMMAND-INPUT-FILE = C-I
  ```

- Default values are provided between slashes, ‘/default/’. The defaults are accepted by hitting Return.
- Real or integer input may be entered irrespective of type of numerical data, use ‘E’ for exponent.
- ‘?’ will list all legal commands and data options. (This command is irrelevant for the graphical user interface where all legal commands and data options are at any time given in the command column of the graphic-mode window.)
- ‘P?’ will list all legal commands starting with P.
- ‘..’ (two dots) will execute the input data before ‘..’ and subsequently abort the current command. The program is thereafter ready for more commands. If the data before the ‘..’ is incomplete it will be discarded.
- ‘,,’ (two commas) will cause one default parameter to be accepted. (May be useful when editing a ‘command input file’.)
- ‘;’ (semicolon) will cause default parameters to be accepted until the end of the parameter group or until there is no default provided.
- Text containing blank characters has to be enclosed within single quotes: 'this is a text'.
- ‘%’ (percentage sign) at the beginning of a line is used for entering a comment. Comments will be logged together with commands on the ‘command log file’ (see Section 4.1.5). Note that the program will occasionally log information on the ‘command log file’, this will appear as comments in between data and comments entered by the user. The program information is preceded by ‘%%’ (two percentage signs) to distinguish it from the user’s own comments. This makes it easy to strip a ‘command log file’ for program information in connection with creating a ‘command input file’ (any fairly good editor will have a macro-functionality or similar enabling you to locate and remove all lines with ‘%%’). Moreover, comments preceded by ‘%%’ will *not* be logged on the ‘command log file’ to avoid irrelevant logging of program information when using an unedited ‘command log file’ as a ‘command input file’.

4.1.5 Files used by Prefem

The file environment of Prefem is illustrated in Figure 4.2. The file extensions (.MOD, .JNL, etc.) are given together with file descriptions.

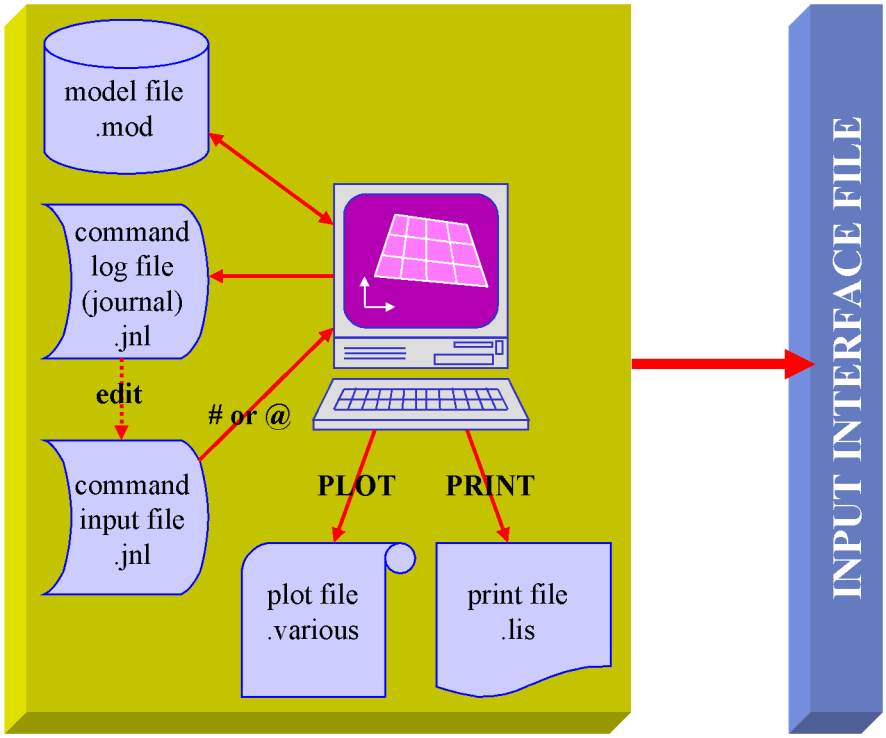


Figure 4.2 The file environment of Prefem

The files are:

- The ‘command log (journal) file’ (.JNL) is an ASCII file on which all commands and data given to the program are logged. This means that both data typed (or clicked) by the user and data read by the program from a ‘command input file’ will be logged. However, commands not changing the model (and data base), e.g. a command displaying data, will not be logged. The time of opening and closing the ‘model file’ is also logged. The file is very useful as a backup file both for verification purposes and for later use as a ‘command input file’. The ‘command log file’ can be read and modified by a text editor.

- The ‘command input file’ (.JNL) is an ASCII file which may be read into the program. The commands contained on this file will have the same effect as if they where given by the user directly. A ‘command input file’ is convenient for batch execution of Prefem; see Section 4.1.7. The file is processed by using the command ‘SET COMMAND-INPUT-FILE ...’ followed by ‘# ALL’ (the latter command means: read all commands found on the file). Alternatively, you may specify a ‘command input file’ when starting Prefem from Manager.

- The ‘model file’ (.MOD) is the binary data base containing all model data. The file cannot be read by a text editor.

- The ‘print file’ (.LIS) is an ASCII file which contains tables over data requested for printing by the PRINT command.

- The ‘plot file’ contains graphic information produced by the PLOT command. The file extension will depend on the plot format chosen (see the SET PLOT FORMAT command). See Section 4.1.6 for advice on using the CGM format to include plots in reports.

- The ‘Input Interface File’ (.FEM) — termed T-file for short — contains the model to be read by a subsequent hydrodynamic or structural analysis program.

Prefem has been designed to protect the user against loss of valuable data. However, accidental loss of data may occur. This may be caused by the user by for example inadvertently deleting the ‘model file’ or it may be due to an inconsistency in the data model. Such inconsistency may occur for several reasons:

- The computer goes down.

- The disk is full, the disk quota is exhausted or user privileges are inadequate.

- There is an error in the program.

If Prefem discovers an inconsistency in the data model the program will normally close all files opened and abort the execution. Prefem may then be restarted using the ‘model file’. In some cases, however, it will not be possible to resume normal execution due to an irrecoverable inconsistency.

If the ‘model file’ is lost it can be reconstructed by re-executing the program and reading input from the ‘command log file’, i.e. using it as a ‘command input file’.

Note: The ‘model file’ will normally not be compatible between different versions of Prefem. The ‘command log file’ may, however, be used as input to a new version.

4.1.6 Creating Plots for Reports

The CGM plot format (see the SET PLOT FORMAT command) is well suited for importing SESAM plots into reports produced by MS Word and other word processors. You may also transfer CGM files from one operating system to another, just make sure to use the ‘binary’ option when transferring the file with FTP (or another protocol).

Depending on the capabilities of your word processor the PostScript plot format may also be used for the purpose of importing SESAM plots into reports. Contrary to CGM, PostScript is an ASCII formatted file and is therefore more easily transferred from one computer make to another.

Note that a word processor will normally recognise only one picture (display) on each file. You should, therefore, specify a new file name for each plot command using the SET PLOT FILE command.

4.1.7 Background Execution

On Unix the user may find it convenient to execute Prefem as a background job rather than as an interactive session. Here is a proposal for how to do this. This proposal is not relevant for executing Prefem through Manager in which case background execution is controlled by Manager.

Execute Prefem in the background as follows:

- Prepare a file (e.g. a revision of a previous ‘command log file’) containing the input data, let the name of the file be FILE_IN.JNL.

- Prepare a file with the following contents (the entries FILE and FILE_IN are example file names):

```
' '
FILE
NEW
SET COMMAND-INPUT-FILE ' ' FILE_IN
..
# ALL
..
EXIT
```

  The two apostrophes in the first and fourth lines enclose a blank space (it may also be a blank line) to specify void prefixes. If a prefix is given in the fourth line, e.g. PREFIX, it will precede the given command input file name requiring the full name of the file containing the input data to be PREFIXFILE_IN.JNL.

- Start Prefem as a background job with the file above as input file.

4.1.8 Command Line Arguments

It is possible to specify command line arguments when starting Prefem. A command line argument will influence the program execution in various ways. On Unix systems the command line arguments are simply added to the command for starting the program:

```
prompt> prefem /NOHEADER/STAT=OLD/INT=LINE/C-F=TEST_IN.JNL/FORCED-EXIT
```

The command line arguments are:

/PREFIX=text	General file name prefix
/NAME=text	General file name
/STATUS=text	Data base / journal file status
/INTERFACE=*LINE*	Start the program in line-mode.
/INTERFACE=PICK	Start the program in graphical user interface mode.
/HEADER=NONE	Do not show the program header.
/NOHEADER	Do not show the program header.
/HEADER=*SHORT*	Show the standard program header.
/WRITE-SUPERELEMENT=number	Write an Input Interface File with the given superelement number when exiting the program.
/*NOWRITE-SUPERELEMENT*	Do not write an Input Interface File.
/COMMAND-FILE=filename	Read the specified command input file after the model/journal file has been accepted.
/*NOCOMMAND-FILE*	Do not read a command input file.

/FORCED-EXIT	Force EXIT after initialisation and after processing of the file defined by the /COMMAND-FILE argument.
/*NOFORCED-EXIT*	Disable FORCED-EXIT.
/EYEDIR-X=value	Set initial eye direction X-value.
/EYEDIR-Y=value	Set initial eye direction Y-value.
/EYEDIR-Z=value	Set initial eye direction Z-value.
/PLOT-FORMAT=format	Set the default plot format to the specified format. For legal alternatives see the SET PLOT FORMAT command.
/PLOT-COLOUR=ON (or *OFF*)	Switch colours for plot file on (or off).
/PLOT-PAGE-SIZE=size	Set the default plot page size.
/PLOT-ORIENTATION=orientation	Set the default plot orientation.
/PRINT-FORMFEED=format	Set the formfeed (page break) character to either ASCII character 12 (format=ASCII) or to the FORTRAN standard of 1 in the first column (format=FORTRAN). ASCII format is default and will give proper page breaks when printing on laser printers and when importing into word processors.
/WINDOW-SIZE=size	Set the size of the graphical user interface window (or graphic display window). This is available on Unix only. The value to give is percentage of screen height. By default size=90.

Note the following about how to enter command line arguments:

- Command line arguments and values can be abbreviated.
- Each argument name must begin with a slash (/) and each argument value must be preceded by an equal sign (=). Spaces can freely be distributed around the equal sign and before each slash.
- Texts with blank spaces and special characters (e.g. file names) must be enclosed in quotes. Note that some operating systems change the case of the input text if it is not enclosed in quotes.
- Slanted arguments or values indicate that these are defaults.
- If at least one of the arguments /PREFIX, /NAME and /STATUS is specified then the prompt for data base and journal file name is skipped and defaults are used for any unspecified values.
- The values given to the /EYEDIR are real values. The default is the Prefem default values. If one of the three are given the other two are set to 0.0 unless specified.
- In some cases a virtual screen larger than the real screen is used, e.g. when a PC through an X-emulator is used as a terminal towards a Unix server. In such cases reduce the /WINDOW-SIZE argument value.

4.2 Program Requirements

4.2.1 Execution Time

The execution time required is negligible for most commands. A few commands, however, will require some CPU and should be used with care on low capacity computers. These are the MESH ALL and any command involving extensive selection of geometry by use of wild-card names.

Note: For large models the DISPLAY and MESH commands may require some time. During processing of these commands you may use Shift+Esc to abort them. This will have no effect on the program execution (apart from incomplete execution of the command in question) and you may continue modelling. This is available on PC only; a similar feature is not available on other operating systems.

4.2.2 Storage Space

The initial size of the data base (prior to any modelling) is about 2 MB. 10-20 MB will be sufficient for most models. Big models may require 100 MB and more.

4.3 Program Limitations

Graphics Devices

The graphical user interface is implemented for OSF/Motif X Window and MS Windows. Under OSF/Motif X Window window stretching is disallowed, use the /WINDOW-SIZE command line argument instead.

Memory

Prefem allocates memory buffers for access to data of the data base file. When using the graphical user interface Prefem will allocate memory for the display.

- **File access buffer**

  The memory is allocated when Prefem is started and the amount is fixed until exiting the program. The amount of memory allocated can be changed by editing the configuration (password) file. To change the amount insert (or modify) the line:

  MSIZE-PREFEM-BUFFER buffer-bytes

  where buffer-bytes represents the amount of memory Prefem will allocate in bytes. The default value is 2457600 (2.4576 millions) representing 150 buffers of 16384 bytes each. The buffer should be changed if, for example, there is not enough memory to use the graphical user interface. Note, however, that increasing the memory for buffers will not improve performance much.

- **Memory for graphical user interface**

  The graphic-mode window will use memory and allocate it when needed. Large displays will need more memory than small displays.

Typing

While typing a command using the keyboard you cannot click commands in menus or select geometry by clicking or use the mouse in any other way until the Return key has been hit or until the typed text has been deleted by backspace.

5 COMMAND DESCRIPTION

The hierarchical structure of the commands and numerical data is documented in this chapter by use of tables. How to interpret these tables is explained below. Examples are used to illustrate how the command structure may diverge into multiple choices and converge to a single choice.

In the example below command A is followed by either of the commands B and C. Thereafter command D is given. Legal alternatives are, therefore, A B D and A C D.

A	B C	D

In the example below command A is followed by three selections of either of commands B and C as indicated by *3. For example: A B B B, or: A B B C, or A C B C, etc.

A	B C	*3

In the example below the three dots in the left-most column indicate that the command sequence is a continuation of a preceding command sequence. The single asterisk indicate that B and C may be given any number of times. Conclude this sequence by the command END. The three dots in the right-most column indicate that the command sequence is to be continued by another command sequence.

...	A	B C	*	...
		END		

In the example below command A is followed by any number of repetitions of either of the sequences B D and C D. Note that a pair of braces ({ }) is used here merely to define a sequence that may be repeated. The braces are not commands themselves.

A	{	B C	D	}*

The characters A, B, C and D in the examples above represent parameters being COMMANDS (written in upper case) and numbers (written in lower case). All numbers may be entered as real or integer values. Brackets ([ ]) are used to enclose optional parameters.

Note: The command END is generally used to end repetitive entering of data. Using double dot (..) rather than END to terminate a command will, depending on at which level in the command it is given, save or discard the data entered. Generally, if the data entered up to the double dot is complete and self-contained the double dot will save the data. If in doubt, it is always safest to leave a command by entering the required number of END commands.

5.1 Selecting Geometry

Selecting geometric entities is relevant in many commands, e.g. for defining properties. The following ways of selecting geometry are available:

- Graphical selection, see Section 3.9.1
- Line-mode command selection, see Section 3.9.2 and below
- Wild-card selection, see Section 3.9.3
- Selection through set, see Section 3.9.4

Below is a complete list of line-mode command alternatives for selecting geometry. In cases where selecting only a certain type of geometry is relevant, e.g. only surfaces when giving shell/plate thickness, then only the relevant type of geometry is available for selection. In many cases, selecting certain geometry will involve selecting also the lower level geometric entities contained in the selected geometry, e.g. selecting a surface when defining boundary conditions involve selecting the lines and points contained in the surface as well.

Note: Whenever geometry is to be selected you may employ any combination of the line-mode commands explained below and graphical means (clicking and dragging the mouse) as explained in Section 3.9.1.

In the command description the texts ‘select-geometry’, ‘select-bodies’, ‘select-surfaces’, ‘select-lines’ and ‘select-points’ should be understood as the following command syntax.

name-of-geometry	
ALL-POINTS-INCLUDED	
ALL-LINES-INCLUDED	
ALL-SURFACES-INCLUDED	
ALL-BODIES-INCLUDED	
GEOMETRY-OF-ELEMENT	element

(	name-of-geometry				*	)
	EXCLUDE					
	INCLUDE					
	ALL-POINTS-INCLUDED					
	ALL-LINES-INCLUDED					
	ALL-SURFACES-INCLUDED					
	ALL-BODIES-INCLUDED					
	GEOMETRY-OF-ELEMENT		element			
	WITH	MATERIAL	material-name			
		SECTION	section-name			
		THICKNESS	lowthick	highthick		
		END				

PARAMETERS:

name-of-geometry	Name of the geometry to select. May also be a wild-card selection, see Section 3.9.3, and name of a pre-defined set, see Section 3.9.4.
ALL-POINTS-INCLUDED	All points of the geometry model are selected.
ALL-LINES-INCLUDED	All lines of the geometry model are selected.
ALL-SURFACES-INCLUDED	All surfaces of the geometry model are selected.
ALL-BODIES-INCLUDED	All bodies of the geometry model are selected.
GEOMETRY-OF-ELEMENT	The geometry to which the given element belongs is selected (the geometry name is logged on the command log file).
(and)	Parentheses enable giving several entries. Remember entering a space on both sides of the parentheses.
EXCLUDE	Exclude geometry names from geometry already selected.
INCLUDE	Include geometry names, this is relevant after an EXCLUDE command in order to counteract the exclusion.
WITH	Of the geometry currently selected (within the current command) only the part with the appropriate characteristics are selected.
MATERIAL	The given material name is the criterion for selection.
SECTION	The given section name is the criterion for selection.

THICKNESS	The lower and upper limit for thickness is the criterion for selection. Functions cannot have been used for specification of the thickness for this option to work.

5.2 Selecting Nodes and Elements

In accordance with the philosophy of Prefem the user will normally only refer to geometric entities. In certain cases, however, selecting nodes and elements directly may be needed, e.g. within the DEFINE SET and DISPLAY commands. The command syntax for selecting nodes and elements is described below. In cases where selecting only elements (and not nodes) is relevant then only the relevant options will be available.

In the command description the texts ‘select-elements’ and ‘select-nodes’ should be understood as the following command syntax.

name-of-geometry			
NODE-NUMBER	node	*	
	END		
NODE-GROUP	node1	node2	nstep
ALL-NODES-INCLUDED			
ELEMENT-NUMBER	element	*	
	END		
ELEMENT-GROUP	elem1	elem2	estep
ALL-ELEMENTS-INCLUDED			
GEOMETRY-OF-ELEMENT	element		
ALL-POINTS-INCLUDED			
ALL-LINES-INCLUDED			
ALL-SURFACES-INCLUDED			
ALL-BODIES-INCLUDED			

(	name-of-geometry					*	)
	EXCLUDE						
	INCLUDE						
	NODE-NUMBER	node	*				
		END					
	NODE-GROUP	node1	node2	nstep			
	ALL-NODES-INCLUDED						
	ELEMENT-NUMBER	element	*				
		END					
	ELEMENT-GROUP	elem1	elem2	estep			
	ALL-ELEMENTS-INCLUDED						
	NEIGHBOUR-ELEMENTS						
	SPECIFIED-ELEMENTS						
	GEOMETRY-OF-ELEMENT	element					
	ALL-POINTS-INCLUDED						
	ALL-LINES-INCLUDED						
	ALL-SURFACES-INCLUDED						
	ALL-BODIES-INCLUDED						
	WITH	MATERIAL	material-name				
		SECTION	section-name				
		THICKNESS	lowthick	highthick			
		END					

PARAMETERS:

name-of-geometry	Name of the geometry to select. May also be a wild-card selection, see Section 3.9.3, and name of a pre-defined set, see Section 3.9.4.
NODE/ELEMENT-NUMBER	Single nodes/elements are selected, terminate selection by END.
NODE/ELEMENT-GROUP	A group of nodes/elements are selected by giving first, last and step.
ALL-NODES/ELEMENTS-INCLUDED	All nodes/elements are selected.
GEOMETRY-OF-ELEMENT	The geometry to which the given element belongs is selected (the geometry name is logged on the command log file).

ALL-POINTS-INCLUDED	All points of the geometry model are selected.
ALL-LINES-INCLUDED	All lines of the geometry model are selected.
ALL-SURFACES-INCLUDED	All surfaces of the geometry model are selected.
ALL-BODIES-INCLUDED	All bodies of the geometry model are selected.
(and)	Parentheses enable giving several entries. Remember entering a space on both sides of the parentheses.
EXCLUDE	Exclude geometry names from geometry already selected.
INCLUDE	Include geometry names, this is relevant after an EXCLUDE command in order to counteract the exclusion.
NEIGHBOUR-ELEMENT	This is a switch that implies selection also of the neighbouring solid elements of a surface selected. For example, the command (NEIGHBOUR-ELEMENT S1) also selects solid elements being neighbours to S1 whereas the command (S1) only selects (shell/membrane/beam) elements within S1.
SPECIFIED-ELEMENT	This is a switch that counteracts the NEIGHBOUR-ELEMENT switch. I.e. it switches back to the normal or default selection mode.
WITH	Of the nodes/elements currently selected (within the current command) only those with the appropriate characteristics are selected.
MATERIAL	The given material name is the criterion for selection.
SECTION	The given section name is the criterion for selection.
THICKNESS	The lower and upper limit for thickness is the criterion for selection. Functions cannot have been used for specification of the thickness for this option to work.

5.3 Functions

Surface thickness and loads (surface pressure, line loads, temperature, concentrated loads, prescribed displacements and prescribed accelerations) can be specified as values varying in space. Rather than giving a single numeric value for a load component a function describing its variation in space may be entered. A function is defined based on a set of pre-defined basic functions, mathematical functions, arithmetic operators and constants. The various basic and mathematical functions are listed below and described in more detail later in this section.

The basic functions available are:

LINEAR-2POINTS-VARYING, see page 5-10
LINEAR-3POINTS-VARYING, see page 5-11

LINEAR-RADIUS-VARYING, see page 5-12

CYLINDRICAL-ANGLE-VARYING, see page 5-13

CYLINDRICAL-RADIUS-VARYING, see page 5-14

The function may be restricted to only having a value between two points, i.e. zero value outside:

VALUE-BETWEEN, see page 5-15

The validity of the function may be restricted to only elements between two points, i.e. *undefined* (not the same as zero) outside:

ONLY-BETWEEN, see page 5-17

In addition, the following mathematical functions are available:

SIN, see page 5-18

COSIN, see page 5-18

ABS, see page 5-20

SIGN, see page 5-20

EXP, see page 5-19

LN, see page 5-19

DIM, see page 5-20

SQRT, see page 5-20

MAX, see page 5-20

MIN, see page 5-20

The constant π is entered by the parameter:

PI

Parentheses may be used for building up expressions; remember that these are commands in themselves and must be separated from other commands and data by blanks:

(

)

The basic and mathematical functions above have parameters. Rather than giving numeric values for these parameters another function may be given. I.e. for all parameters called ‘value’, ‘value-1’, etc. in the following explanations of the basic and mathematical functions a new function may be entered. Furthermore, arithmetric operators may be used to add, subtract, multiply and divide functions. In this way functions of practically unlimited complexity may be specified. The arithmetric operators are:

+

-

*

/

**

Normal operator precedence is applied, i.e. ** is calculated first, thereafter * and / while + and - are calculated last.

A function is in itself unlimited in space. Its value in any given point in space (in a node in the FE model) is found by projecting the point into the function and evaluating the function there. How to project a point into a function depends on the function and is explained in the following for each function.

Section 3.7 shows a few examples of load application by using functions. Figure 5.1 through Figure 5.5 show a few more examples of functions. The functions in the examples are explained in detail later in this section.

A linear variation along a line is defined by:

```
LINEAR-2POINTS-VARYING P1 -6.4 P2 3.1
```

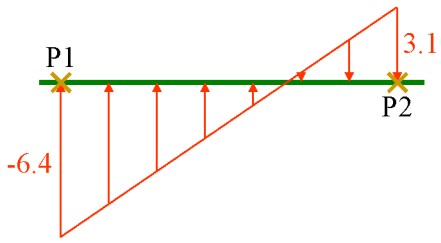


Figure 5.1 Example of linear function

A parabolic variation along a line is defined by:

```
( LINEAR-2POINTS-VARYING
  P1 -6.4 P2 3.1 ** 2.0 )
```

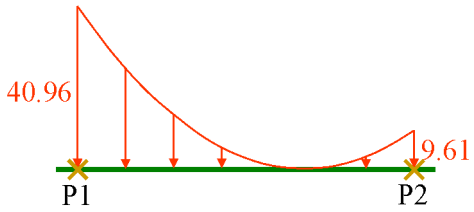


Figure 5.2 Example of parabolic function

A linear function along a circle (linear with respect to ϕ) may be defined by (PA1 and PA2 defines the axis):

```
CYLINDER-ANGLE-VARYING PA1 PA2 P1 1. P2 0.
```

Notice that the function is extrapolated beyond P1 and P2 if necessary to cover all geometry to which the load applies.

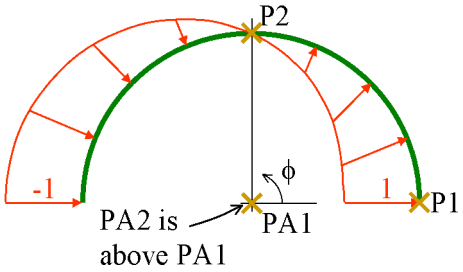


Figure 5.3 Example of linear function along circle

A sinus function along a circle may be defined in two alternative ways: (1) as a sinus function defined along the arc from P1 to P2 and (2) by taking advantage of the fact that a linear function projected onto a circle as shown in the figure will form a sinus function.

```
(1) SIN CYLINDER-ANGLE-VARYING PA1 PA2
    P1 ( PI / 2 ) P2 0.

(2) LINEAR-2POINTS-VARYING P1 1. PA1 0.
```

In (1) notice how the value π/2 is given for P1 using the built-in parameter PI and with spaces separating the left and right parentheses and division symbol (slash).

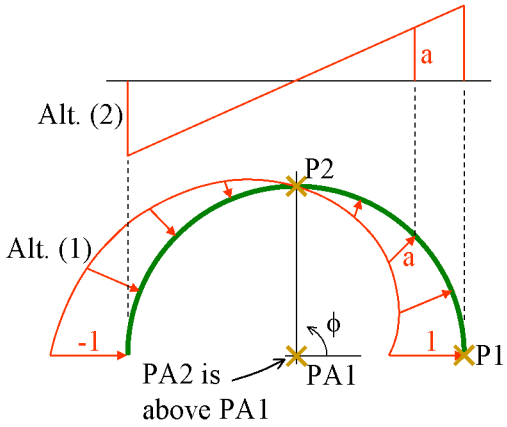


Figure 5.4 Example of sinus function along circle

A wave force is an exponential function of depth. A wave force on a cylinder may be given by multiplying the functions above by:

```
EXP LINEAR-2POINTS-VARYING PA2 0.0 PA1 -1.5
```

Multiplication of functions is performed by enclosing them in parentheses (let 'f' and 'g' be the two functions):

```
( f * g )
```

Note that without the left parenthesis the program will recognise 'f' as a complete expression and not allow continuation of the expression.

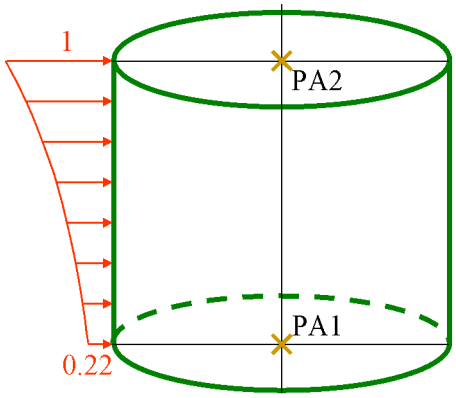


Figure 5.5 Example of exponential function

LINEAR-2POINTS-VARYING

...	LINEAR-2POINTS-VARYING	point-1	value-1	point-2	value-2

PURPOSE:

The command defines a linearly varying value interpolated on the line between two points.

The value at any point on the line between the two points, and the extension of this line, will apply to all points in a plane perpendicular to the line and through the point in question. Also see Figure 3.55 for an illustration of this.

PARAMETERS:

point-1	Name of the first interpolation point
value-1	Value for the first interpolation point
point-2	Name of the second interpolation point
value-2	Value for the second interpolation point

Example of use; see Figure 5.6:

```
LINEAR-2POINTS-VARYING P1 2.5 P2 1.
```

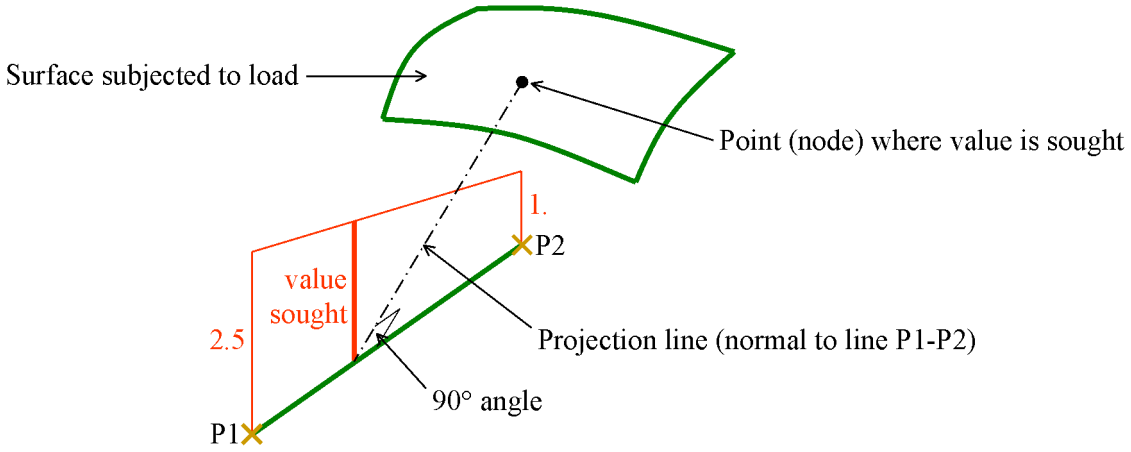


Figure 5.6 Example of use of LINEAR-2POINT-VARYING function

LINEAR-3POINTS-VARYING

...	LINEAR-3POINTS-VARYING	point-1	value-1	point-2	value-2	point-3	value-3

PURPOSE:

The command defines a value varying linearly in two directions. Three points define a projection plane. The function can be viewed as a plane set off from the projection plane by the three values in the three points.

The value at any point in the projection plane, and the extension of this plane, will apply to any point on a line perpendicular to the plane and through the point in question.

PARAMETERS:

point-1	Name of the first interpolation point
value-1	Value for the first interpolation point
point-2	Name of the second interpolation point
value-2	Value for the second interpolation point
point-3	Name of the third interpolation point
value-3	Value for the third interpolation point

Example of use; see Figure 5.7:

```
LINEAR-3POINTS-VARYING P1 3. P2 2.5 P3 1.
```

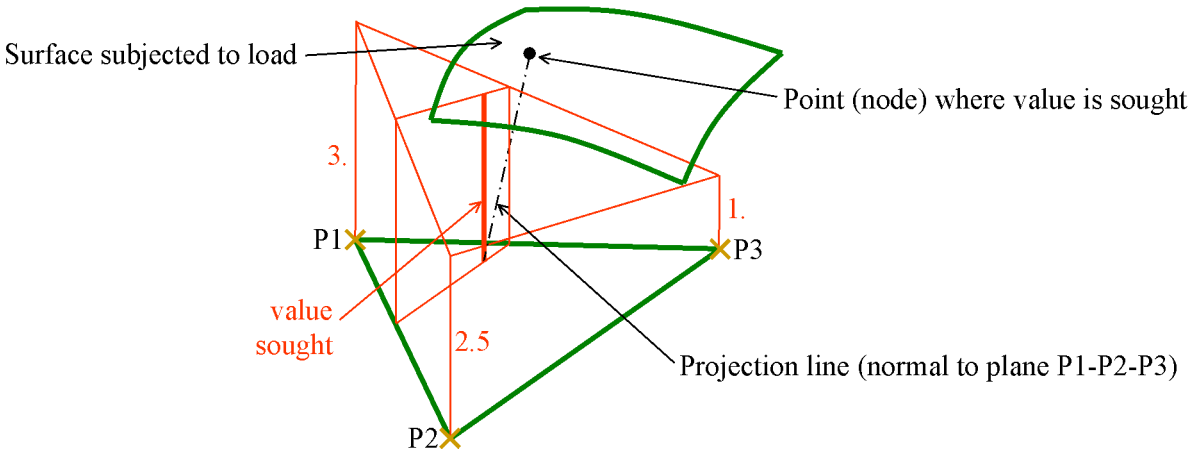


Figure 5.7 Example of use of LINEAR-3POINTS-VARYING function

LINEAR-RADIUS-VARYING

...	LINEAR-RADIUS-VARYING	centre-point	point-1	value-1	point-2	value-2

PURPOSE:

The command defines a value varying linearly with the radius about a given centre point. The value is specified in two points. One of the points can be the same as the centre point. The value will be constant on any sphere with its centre at the centre point.

PARAMETERS:

centre-point	Name of the centre point
point-1	Name of the first point defining the linear value
value-1	Value for the first point
point-2	Name of the second point defining the linear value
value-2	Value for the second point

Example of use; see Figure 5.8:

```
LINEAR-RADIUS-VARYING PC P1 1.3 P2 2.
```

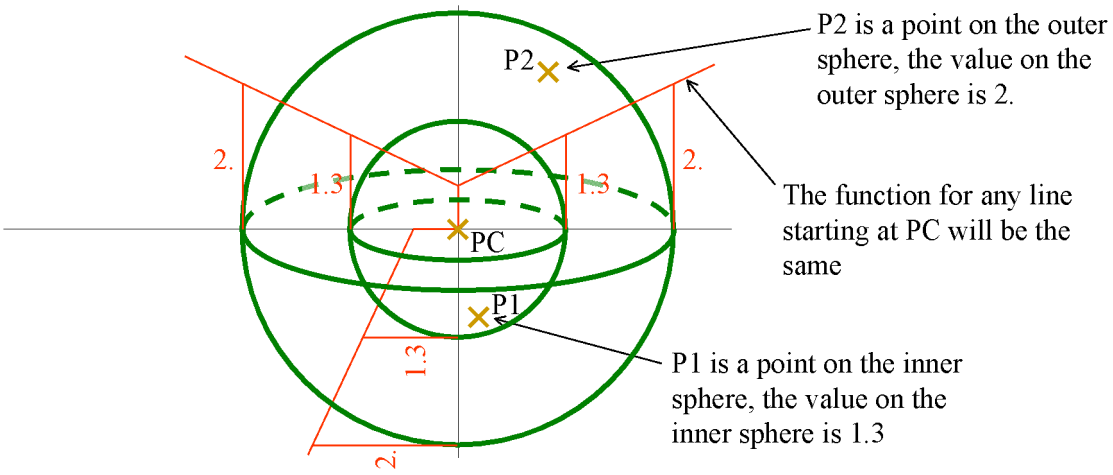

P2
P2 is a point on the outer sphere, the value on the outer sphere is 2.
2.
1.3
1.3
2.
PC
The function for any line starting at PC will be the same
P1
P1 is a point on the inner sphere, the value on the inner sphere is 1.3
1.3
2.

Figure 5.8 Example of use of LINEAR-RADIUS-VARYING function

CYLINDRICAL-ANGLE-VARYING

...	CYLINDER-ANGLE-VARYING	axis-1	axis-2	point-1	value-1	point-2	value-2

PURPOSE:

The command defines a value varying linearly with the angle around a given axis. The value is specified in two points. Neither of the points can be on the axis and a line through the points cannot intersect the axis.

The function can be explained as follows: Each of the two points define a radial plane about the axis. These two radial planes form an angle. The value will vary linearly about this angle starting in the radial plane defined by the first point (point-1) and extrapolated up to 180°. Past the 180° angle the value will vary linearly back to the given value in the radial plane defined by the first point. The value will be constant in any radial plane.

PARAMETERS:

axis-1	Name of the first point defining the axis
axis-2	Name of the second point defining the axis
point-1	Name of the first point defining the linear value
value-1	Value in the first point
point-2	Name of the second point defining the linear value
value-2	Value in the second point

Example of use; see Figure 5.9:

```
CYLINDER-ANGLE-VARYING AP1 AP2 P1 -3.9 P2 -2.1
```

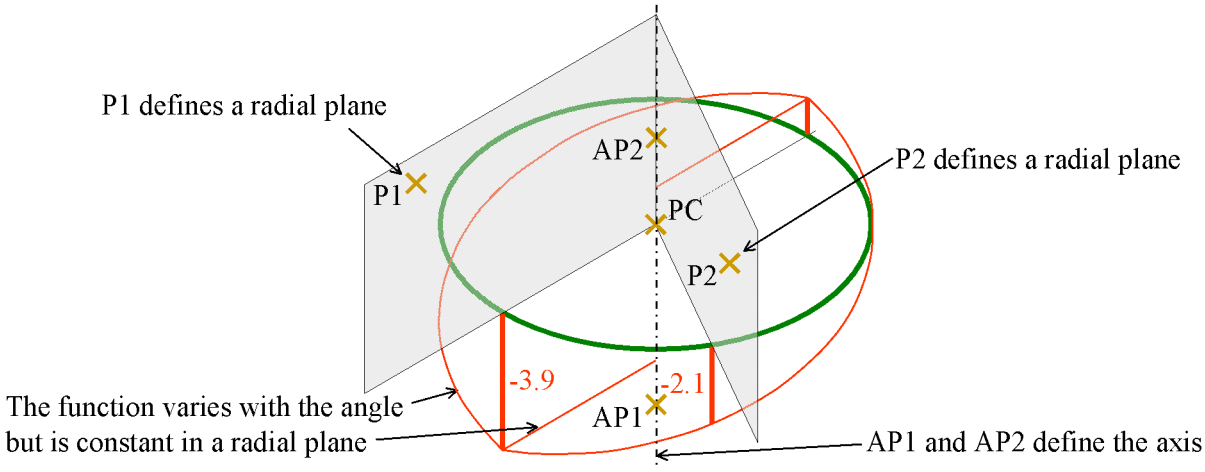


Figure 5.9 Example of use of CYLINDER-ANGLE-VARYING function

CYLINDRICAL-RADIUS-VARYING

...	CYLINDER-RADIUS-VARYING	axis-1	axis-2	point-1	value-1	point-2	value-2

PURPOSE:

The command defines a value varying linearly with the distance from a given axis. The value is specified in two points. One of the points can lie on the axis. The value will be constant on any cylinder around the axis.

PARAMETERS:

axis-1	Name of the first point defining the axis
axis-2	Name of the second point defining the axis
point-1	Name of the first point defining the linear value
value-1	Value in the first point
point-2	Name of the second point defining the linear value
value-2	Value in the second point

Example of use; see Figure 5.10:

```
CYLINDER-RADIUS-VARYING AP1 AP2 P1 1.3 P2 2.
```

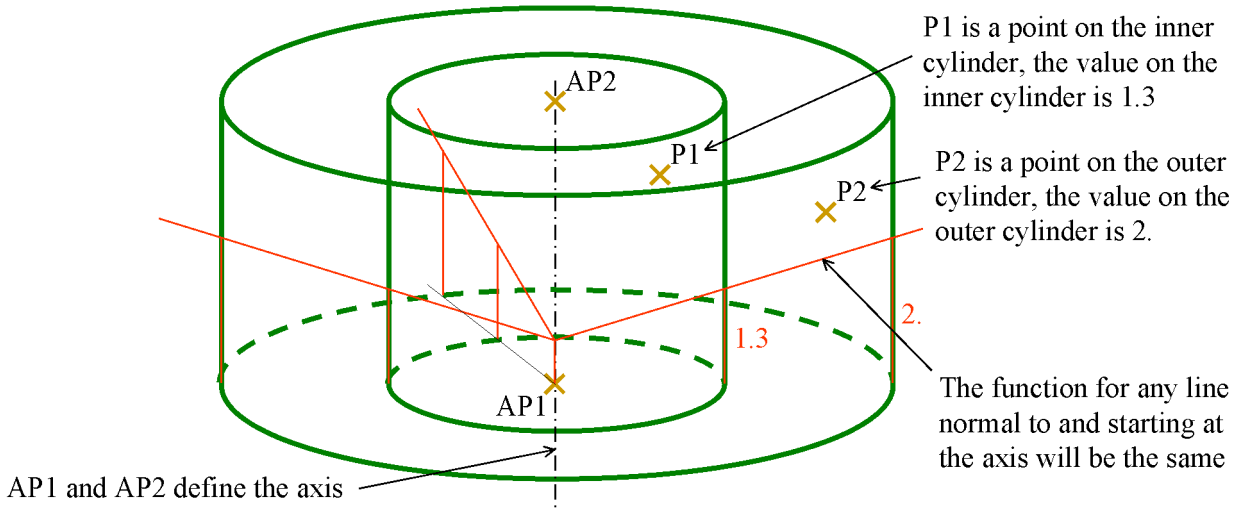

AP2
P1
P2
2.
1.3
AP1
P1 is a point on the inner cylinder, the value on the inner cylinder is 1.3
P2 is a point on the outer cylinder, the value on the outer cylinder is 2.
The function for any line normal to and starting at the axis will be the same
AP1 and AP2 define the axis

Figure 5.10 Example of use of CYLINDER-RADIUS-VARYING function

VALUE-BETWEEN

...	VALUE-BETWEEN	point-1	point-2	value

PURPOSE:

The command limits the application of a value (or function) to a specified space. Outside the space the value will be zero.

The application is limited to the space in between the two parallel planes normal to the line between the two points and through the points.

Note: Be aware of the difference between this function and ONLY-BETWEEN by studying the examples given for each.

Note: When used to describe a line-load for a two node beam element the interpretation of this function is somewhat different. The reason for this is that a two node beam element — contrary to a membrane/shell/volume element — can have a discontinuity of the load. Such discontinuity is described by the VALUE-BETWEEN function. Compare Figure 5.11 and Figure 5.12 to see the difference.

PARAMETERS:

point-1	Name of the first limiting point
point-2	Name of the second limiting point
value	Value (or function)

Example of use; see Figure 5.11 for membrane/shell/solid elements and Figure 5.12 for 2 node beam element:

```
VALUE-BETWEEN P1 P2 1.2
```

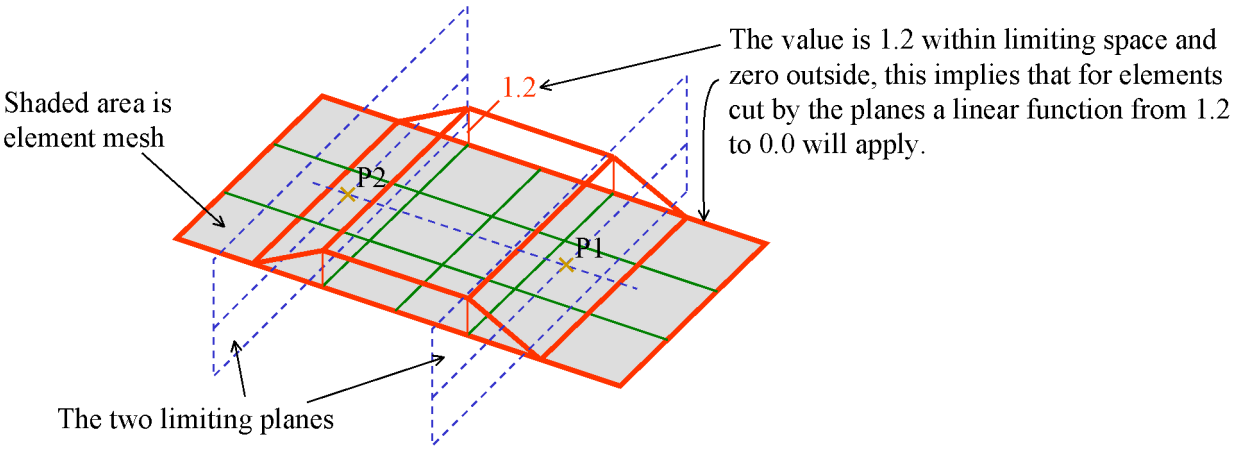

Shaded area is
element mesh
1.2
The value is 1.2 within limiting space and
zero outside, this implies that for elements
cut by the planes a linear function from 1.2
to 0.0 will apply.
P2
P1
The two limiting planes

Figure 5.11 Example of use of VALUE-BETWEEN function for membrane/shell/solid elements

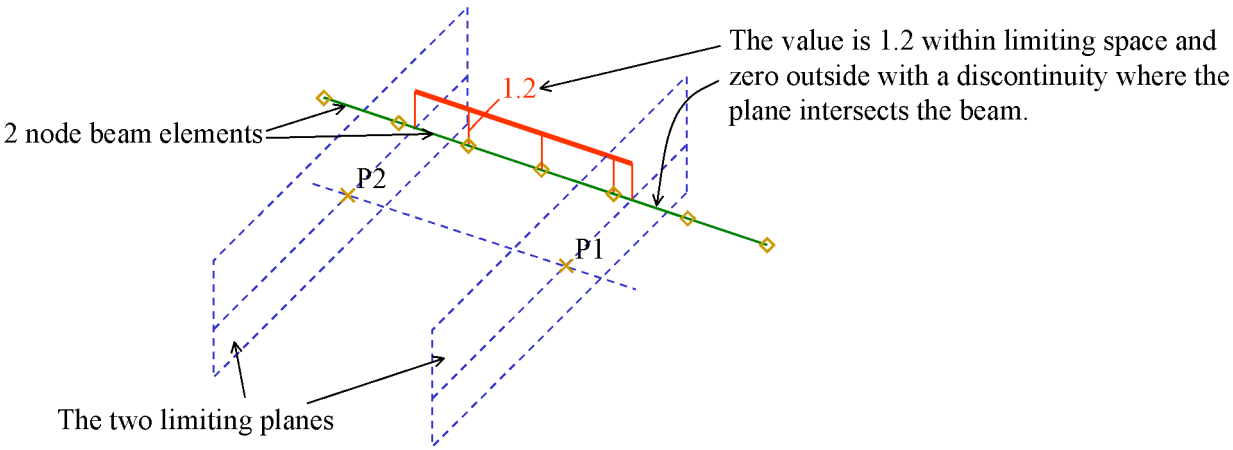


Figure 5.12 Example of use of VALUE-BETWEEN function for 2 node beam element

ONLY-BETWEEN

...	ONLY-BETWEEN	point-1	point-2	value

PURPOSE:

The command limits the application of a value (or function) to a specified space. Outside the space no value is defined (not the same as zero, see below).

This function is not suitable for specifying variable thickness.

The application is limited to the space in between the two parallel planes normal to the line between the two points and through the points.

Note: Be aware of the difference between this function and VALUE-BETWEEN by studying the examples given for each.

PARAMETERS:

point-1	Name of the first limiting point
point-2	Name of the second limiting point
value	Value (or function)

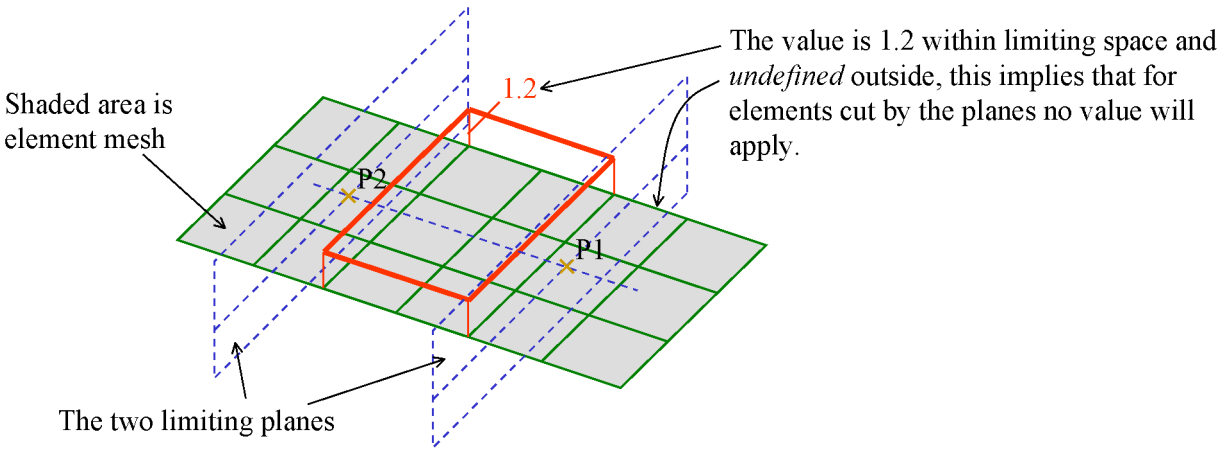

Shaded area is element mesh
1.2
The value is 1.2 within limiting space and *undefined* outside, this implies that for elements cut by the planes no value will apply.
P2
P1
The two limiting planes

Figure 5.13 Example of use of ONLY-BETWEEN function

SIN / COSIN

...	SIN	argument
	COSIN	

PURPOSE:

The command calculates sinus and cosine of the argument. The argument is in radians.

Simple sinus and cosine functions are established by using the LINEAR-2POINTS-VARYING function as argument. (This linear function is Prefem's way of establishing the argument 'x' in the sinus and cosine functions 'sin x' and 'cos x'). See the example below.

PARAMETERS:

argument Argument (or function)

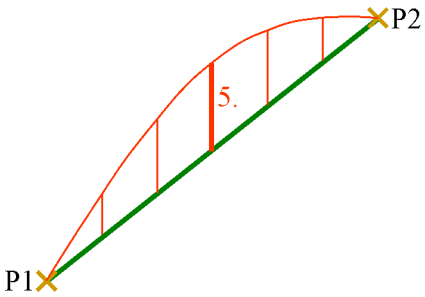


Figure 5.14 Example of use of SIN function

EXP / LN

...	EXP	argument
	LN	

PURPOSE:

EXP calculates the exponential function (e^x).

LN calculates the natural logarithm (ln x).

An exponential function and a natural logarithm function are established by using the LINEAR-2POINTS-VARYING function as argument. See the explanation for the SIN/COSIN functions for how to use the EXP and LN functions.

PARAMETERS:

argument Argument (or function)

ABS / SIGN / MAX / MIN / DIM / SQRT

...	ABS	argument	
	SIGN		
	MAX	argument1	argument2
	MIN		
	DIM		
	SQRT	argument	

PURPOSE:

ABS calculates the absolute value of the argument.

SIGN determines the sign of the argument as follows:

- (SIGN a) = 1 if a > 0
- (SIGN a) = -1 if a < 0
- If a = 0 (or ≈ 0) then the function may give -1, 0 or 1 as result depending on the numerical accuracy.

MAX / MIN determines the maximum / minimum of the two arguments. For example:

- (MAX a1 a2) = a1 if a1 > a2
- (MIN a1 a2) = a2 for the same condition.

DIM calculates the positive difference between the two arguments as follows:

- (DIM a1 a2) = a1 - a2 if a1 > a2
- (DIM a1 a2) = 0 if a1 < a2

SQRT calculates the square root of the argument. A square root function is established by using the LINEAR-2POINTS-VARYING function as argument. See the explanation for the SIN/COSIN functions for how to use the SQRT function. The square root function is equivalent to giving the function ‘argument ** 0.5’.

PARAMETERS:

argument	Argument (or function)
argument1	Argument (or function)
argument2	Argument (or function)

5.4 Detailed Description of Commands

The input commands are described in the following. The commands and subcommands are described in alphabetic order. Below is a list of all main (basic level) commands.

ADD-DISPLAY See page 5-23.

CHANGE See page 5-25.

CHECK See page 5-40.

CONNECT See page 5-47.

COPY See page 5-48.

CREATE See page 5-51.

CUT See page 5-54.

DEFINE See page 5-56.

DELETE See page 5-92.

DISPLAY See page 5-96.

EXIT See page 5-98.

EXTRUDE See page 5-99.

GENERATE See page 5-102.

HELP See page 5-105.

JOIN See page 5-106.

LABEL See page 5-107.

LOCATE See page 5-112.

MESH See page 5-113.

PLOT See page 5-115.

PRINT See page 5-117.

PROPERTY See page 5-126.

RE-COMPUTE See page 5-206.

RE-DISPLAY See page 5-207.

READ See page 5-208.

ROTATE See page 5-211.

SET See page 5-212.

WRITE See page 5-262.

ZOOM See page 5-263.

See page 5-264.

ADD-DISPLAY

ADD-DISPLAY	NODE	select	
	ELEMENT		
	SHAPE		
	POINT		
	LINE		
	SURFACE		
	BODY		
	GEOMETRY		
	FIND	geometry-name	
	MESH		
	LOAD	load-case	load-type

PURPOSE:

The command adds element mesh, geometry and load display to the current display without refreshing the screen. Typically, the mesh may be added to the geometry display, shapes (tools for modelling) may be added and load display may be added to the mesh.

Also, the command highlights a named geometric entity (the FIND option). This is used to locate a single point, line, surface or body in a geometry display.

PARAMETERS:

NODE — Add selected nodes to the current display.

ELEMENT — Add selected elements to the current display.

SHAPE — Add selected shapes (see the DEFINE SHAPE command) to the current display. A shape is in itself unlimited in space (except for the spherical shape) but it is displayed limited by the points defining it.

POINT — Add selected points to the current display.

LINE — Add selected lines to the current display.

SURFACE — Add selected surfaces to the current display.

BODY — Add selected bodies to the current display.

select — The parts to be added to the display. This may be a selection of geometry, see Section 5.1, a selection of nodes or elements, see Section 5.2, or a selection of shapes. Selecting shapes is done in a similar way as selecting geometry.

GEOMETRY	Add the geometry to the current display. (This command cannot be used after a DISPLAY MESH command.) Only the geometry related to the currently displayed elements/nodes will be added.
FIND	Highlight a single geometric entity.
geometry-name	The name of a single geometric entity (point, line, surface or body).
MESH	Add the mesh to the current display. Only the mesh related to the currently displayed geometry will be added.
LOAD	Add a selected load display to the current display. The load is presented as arrows with their heads in the nodes where the load applies. The arrow lengths are proportional with the magnitude of the load. The largest of the arrows will have a length on the display of approximately 15 mm (this length may be adjusted by the SET GRAPHICS SIZE-SYMBOLS LOAD-ARROW-SIZE command). The tails of the arrows are connected by dotted lines (the SET GRAPHICS PRESENTATION LOADS command removes these dotted lines). Only the part of the load related to the currently displayed elements will be added. The positions of the arrows are adjusted for shrunken elements, for element thickness shown and for eccentricity of beam elements. It is thus possible to distinguish between line loads on shell elements and line loads on beam elements even when they are applied in the same position.
load-case	Load case to add. If a load case contains several load types (line load, normal pressure, etc.) then only one of the load types may be added at a time. (You may add more load types and even more load cases to the same display on the screen but once you use RE-DISPLAY or make a plot only the last load case and type will appear.)
load-type	Type of load to add. See the PROPERTY LOAD command for the different types of loads. In addition to specifying a specific type of load the option ALL-LOAD-TYPES may be given to display all types of loads at once. Plotting of ALL-LOAD-TYPES is, however, not possible. See under NOTES below.

NOTES:

If you change the viewing position (e.g. by SET GRAPHICS EYE-DIRECTION or ROTATE) you should do a RE-DISPLAY before you use the ADD-DISPLAY command.

If you use the ALL-LOAD-TYPES option to display loads then the displayed loads will disappear once you rotate or zoom the model. Furthermore, a plot will not include the displayed loads.

CHANGE

CHANGE	ARC	...
	BODY	
	CRACK[a]	
	DAMPER	
	ELEMENT-ATTRIBUTE	
	INTERSECTION	
	LINE	
	MASS-ELEMENT	
	MESH	
	NAME	
	NODE	
	NODE-LINE	
	NORMAL-OF-SURFACE	
	POINT	
	PRISM	
	PROPERTY	
	ROTATION-OF-SURFACE	
	SET	
	SHAPE	
	SPLINE	
	SPRING	
	SUPERELEMENT[b]	
	SURFACE	
	TRANSFORMATION	

a. This option is presently inactive.
b. This option is presently inactive.

PURPOSE:

The command changes parameters or values of previously defined geometry, properties and other data. Most of the commands have the same syntax and corresponding interpretation as the commands defining the data. Therefore, rather than describing these commands in detail here reference is made to the corresponding DEFINE or PROPERTY commands.

However, some CHANGE commands demand special explanations, these are found on the following pages: These are:

- CHANGE ARC / INTERSECTION / LINE / SPLINE / NODE-LINE
- CHANGE ELEMENT-ATTRIBUTE
- CHANGE MESH
- CHANGE NAME
- CHANGE NODE
- CHANGE NORMAL-OF-SURFACE / ROTATION-OF-SURFACE
- CHANGE POINT
- CHANGE PROPERTY LOAD load-case TO-MASS

NOTES:

If changes are made to loads by the CHANGE PROPERTY LOAD command then use the RE-COMPUTE LOADS command to redistribute the loads, otherwise an ADD-DISPLAY LOAD command will not give correct result. This re-computation will automatically be performed when producing the Input Interface File.

Changes made to a transformation using the CHANGE TRANSFORMATION command will be added to its current definition. Such a change will thus have the same effect as if the data was entered at the end of the command that defined the transformation. See the DEFINE TRANSFORMATION command (or the PROPERTY TRANSFORMATION command).

Rather than using the CHANGE PROPERTY TRANSFORMATION command the CHANGE TRANSFORMATION command is recommended as the latter is the more flexible and powerful one.

Wild-card selection of geometry cannot be done for several of the CHANGE commands.

CHANGE ARC / INTERSECTION / LINE / SPLINE / NODE-LINE

See the DEFINE command for the command syntax.

PURPOSE:

The command changes the definition of lines/curves.

Any of the commands CHANGE ARC / INTERSECTION / LINE / SPLINE / NODE-LINE can be used for changing any line and curve defined by the DEFINE ARC / INTERSECTION / LINE / SPLINE / NODE-LINE or the GENERATE commands. This is done by using e.g. the CHANGE ARC command for a line defined by DEFINE LINE. The line/curve then changes type as well as geometry (the points defining it). Such a change can be done even when the line/curve is one of the borderlines of a surface on condition that the start and end points remain the same. Also see Section 3.3.1. Note that wild-card notation cannot be used for changing several lines/curves.

Note: A curve resulting from using the GENERATE command in cylindrical or spherical coordinate systems cannot be changed directly into a straight line using the CHANGE LINE command. This is because the curve is already considered to be a 'line' in the cylindrical/spherical coordinate system. This implies that using the CHANGE LINE command for such a curve will change it into another 'line' in the cylindrical/spherical coordinate system, i.e. a curve. A curve may, however, be changed to an arc or a spline and this arc or spline may in turn be changed into a straight line. Figure 5.15 exemplifies this.

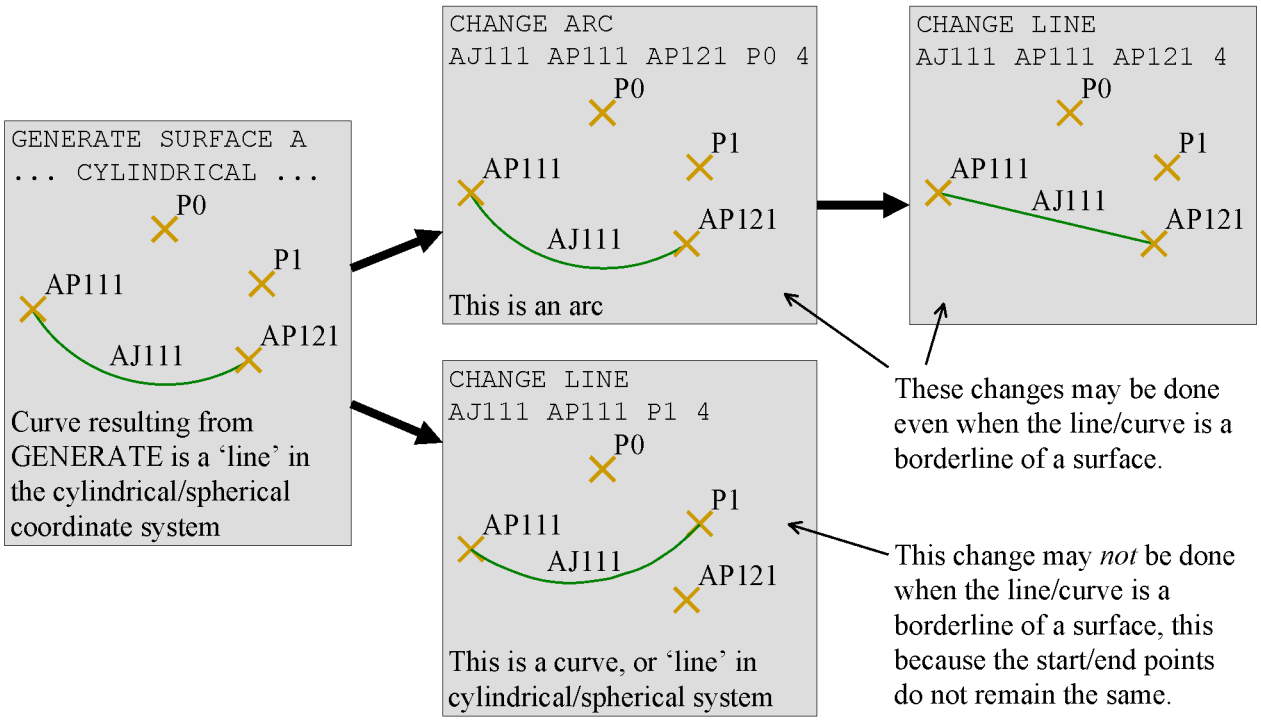

GENERATE SURFACE A
... CYLINDRICAL ...
P0
P1
AP111
AJ111
AP121
Curve resulting from GENERATE is a 'line' in the cylindrical/spherical coordinate system
CHANGE ARC
AJ111 AP111 AP121 P0 4
This is an arc
CHANGE LINE
AJ111 AP111 P1 4
This is a curve, or 'line' in cylindrical/spherical system
CHANGE LINE
AJ111 AP111 AP121 4
These changes may be done even when the line/curve is a borderline of a surface.
This change may *not* be done when the line/curve is a borderline of a surface, this because the start/end points do not remain the same.

Figure 5.15 Changing a curve resulting from GENERATE in cylindrical/spherical systems

Note: Lines/curves will also change by changing the coordinates of the points defining the lines/ curves, i.e. using the CHANGE POINT command.

Note: A curve resulting from using the GENERATE command in a cylindrical or spherical coordinate system will remain a curve in these systems after changing the coordinates of the start or end points.

CHANGE ELEMENT-ATTRIBUTE

...	ELEMENT-ATTRIBUTE	select-geometry	graphic-select-elem	MATERIAL	material-name
				THICKNESS	thickness

PURPOSE:

The command changes thickness and material for selected elements. While the normal procedure for assigning thickness and material is to assign such attributes to geometry (i.e. all elements within the selected geometry) this command allows giving thickness/material to specific elements within, for example, a surface.

The procedure is as follows:

- First select the geometry to which the elements belong, the elements of the selected geometry are then displayed.
- Then select elements using the graphic rubberband (or polygon). The selected elements are then displayed.
- Finally, modify the element thickness or material.

Note: If the mesh is deleted and re-created the changes will be lost. The CHANGE ELEMENT-ATTRIBUTE command must then be repeated.

Note: Several geometric entities may be selected and the subsequent graphic selection may bridge these geometric entities.

Note: Several surfaces overlapping in the display (one behind the other) may be selected (for example using rubberband) followed by a rubberband/polygon selection of elements. The corresponding elements in all the surfaces will then be selected.

The element selection of the command is logged in terms of screen-coordinates of the graphic rubberband (or polygon) and not in terms of element numbers. This involves that when the log file is used as command input file in a new session the elements in the same positions within the same surfaces will be selected even when the element numbers have changed due to a different meshing sequence or a new meshing algorithm.

Note: Referring to the paragraph above: When a log file is used as command input file in a new session and a different mesh is created for the area in question then the new elements may and may not fall within the graphically selected area thereby producing unexpected results.

PARAMETERS:

select-geometry — Select geometry containing elements to be changed. See Section 5.1 on how to perform a selection.

graphic-select-elem — Select elements to be changed by dragging a rubberband completely enclosing them.

MATERIAL	The material of the elements is to be changed.
material-name	The new material name which must have been defined.
THICKNESS	The thickness of the elements is to be changed.
thickness	The new thickness.

CHANGE MESH

...	MESH	select-surfaces	DISPLAY	select-and-move-node

PURPOSE:

The command changes the mesh by moving a selected node.

The command is designed to be used in graphic mode but is still logged for re-use in line-mode (through a command input file).

Move a node as follows:

- Select the surface where nodes shall be moved. Several identical surfaces (identical geometry and identical mesh) may be selected provided that they are displayed on top of each other (seen from an angle so that they overlap).
- In graphic mode give DISPLAY (optionally followed by the Direct access button 'Zoom Fr') to display the mesh of the selected surface. The view angle should preferably be normal to the mesh, rotate the display if necessary.
- Position the cursor over the node you want to move and press and hold the left mouse button (LMB). Drag to the new position of the node and release the LMB. (The node will be moved the distance the mouse is moved.)
- The display will be refreshed with the new node position and another node may be moved.

The following rules apply to moving nodes:

- If the node is in a point it will *not* be moved.
- If the node is on a line it will be moved but only along the line.
- If the node is on a surface projected onto a shape it will be moved and then projected back onto the shape.
- If the node is on a surface not projected onto a shape then three nodes from an element to which the node belongs will be used to form a plane. The node will be projected back onto this plane.

Note: This command may give incorrect results for nodes belonging to solid elements.

Note: If the node is on a surface with a cylindrical or spherical coordinate system assigned to it, see the SET MESH command, (i.e. the surface is not projected onto a shape) the node may be moved off the surface.

Note: It may prove difficult to improve a mesh manually by this command. You may find that the only way to get an acceptable mesh is to change the data determining the mesh, i.e. the number of elements or maximum element size for lines and mesh corners.

Note: No mesh smoothing will be performed. E.g. mid-side nodes of eight node shell elements will not be moved when corner nodes are moved.

PARAMETERS:

select-surfaces	Surfaces for which nodes shall be moved. See Section 5.1 on how to perform a selection.
DISPLAY	Display the mesh of the selected surface(s) only. Note that this command will not be logged. It has meaning for graphic-mode only.
select-and-move-node	Position the cursor over the node you want to move and press and hold the left mouse button (LMB). Drag to the new position of the node and release the LMB. This action will be logged as three sets of coordinates: The first point and second point in space on a line normal to the screen identifying the node to be moved and the distance in space by which the node is moved. Note that entering the three sets of coordinates manually is awkward and of little practical interest.

CHANGE NAME

...	NAME	select-geometry	new-names

PURPOSE:

The command changes the name of selected geometry.

PARAMETERS:

select-geometry — Select geometry to be renamed. See Section 5.1 on how to perform a selection.

new-names — The new name of the selected geometry. If more than one geometric entity has been selected (either by use of parentheses or wild-card selection) then wild-card construction of the new names must be used.

EXAMPLES:

The following command will change the names AP111 to BP111 and AJ111 to BJ111.

```
CHANGE NAME ( AP111 AJ111) B*
```

The following command will change all names starting with A to corresponding names starting with B.

```
CHANGE NAME A* B*
```

CHANGE NODE

...	NODE	node-number	x	y	z

PURPOSE:

The command changes the coordinates of a node. The shape of the adjoining elements will consequently change.

PARAMETERS:

node-number	Number of the node.
x	New X-coordinate. The old coordinate is used as the default value.
y	New Y-coordinate. The old coordinate is used as the default value.
z	New Z-coordinate. The old coordinate is used as the default value.

NOTES:

This command can only be performed after creating the mesh. If the mesh is deleted and recreated then the result of a CHANGE NODE command is lost.

If variable thickness of surfaces is defined then the PROPERTY THICKNESS command must be used after the CHANGE NODE command or else the node may have incorrect thickness (it will retain its thickness from the original position).

CHANGE NORMAL-OF-SURFACE / ROTATION-OF-SURFACE

...	NORMAL-OF-SURFACE	+X-GLOBAL-INFINITY				select-surfaces
		+Y-GLOBAL-INFINITY				
		+Z-GLOBAL-INFINITY				
		-X-GLOBAL-INFINITY				
		-Y-GLOBAL-INFINITY				
		-Z-GLOBAL-INFINITY				
		FROM-FIXED-POINT	x	y	z	
		GUIDING-POINT-DIRECTION				
		TOWARDS-FIXED-POINT				
	ROTATION-OF-SURFACE					

PURPOSE:

These two commands have the same purpose: To set the direction of the local z-axis of shell/membrane elements. The NORMAL-OF-SURFACE option sets the z-axes explicitly while the ROTATION-OF-SURFACE option reverses the current directions.

When a surface is defined the borderlines are sorted in a certain sequence. This rotational sequence defines by the right-hand-rule a surface normal which in turn determines the element z-axes. The direction of the z-axis of an element determines the direction of the normal pressure type of loading (this may be overruled though, see the SET INSIDE / OUTSIDE command). The z-axis direction is also of consequence for the interpretation of results: if two neighbouring and co-planar surfaces have opposite normals then the results are easily misinterpreted. Setting or changing the surface normal may thus be necessary.

In the NORMAL-OF-SURFACE option the direction of the surface normal is defined by a guiding point. This guiding point may be defined by giving its coordinates directly or by positioning it infinitely far away along any of the global axes, either in positive or in negative direction. The guiding point lies on the positive surface normal side. This is, however, not true for the FROM-FIXED-POINT alternative which involves that the guiding point lies on the negative surface normal side.

PARAMETERS:

NORMAL-OF-SURFACE — Set the surface normal explicitly.

+X-GLOBAL-INFINITY — The guiding point is positioned infinitely far out along the positive global X-axis.

+Y-GLOBAL-INFINITY — The guiding point is positioned infinitely far out along the positive global Y-axis.

+Z-GLOBAL-INFINITY — The guiding point is positioned infinitely far out along the positive global Z-axis.

-X-GLOBAL-INFINITY	The guiding point is positioned infinitely far out along the negative global X-axis.
-Y-GLOBAL-INFINITY	The guiding point is positioned infinitely far out along the negative global Y-axis.
-Z-GLOBAL-INFINITY	The guiding point is positioned infinitely far out along the negative global Z-axis.
FROM-FIXED-POINT	The guiding point is a fixed point positioned on the negative surface normal side.
GUIDING-POINT-DIRECTION	The guiding point is positioned at a given vector away from the first point of the surface.
TOWARDS-FIXED-POINT	The guiding point is a fixed point positioned on the positive surface normal side.
x y z	The cartesian coordinates of the guiding point or the vector given in the cartesian coordinate system.
ROTATION-OF-SURFACE	Reverse the current surface normal.
select-surfaces	Surfaces for which the element normal will be changed. See Section 5.1 on how to perform a selection.

CHANGE POINT

...	POINT	points	x	y	z	
			<	*update-coordinates*		>

PURPOSE:

The command changes the coordinates of one or more points. The lines defined based on these points will also change.

Coordinate values can be specified explicitly or by an *update-coordinates* mode. This mode is entered into and concluded by the arrow bracket (less than and greater than) commands '<' and '>'.

The coordinates of several points may be changed by selecting points by wild-card selection. The *update-coordinates* mode may then be used to for example add or subtract a constant value to either of the coordinates of the points.

PARAMETERS:

points	Select points — wild-card, parentheses and graphical means may be used to select several points. Selecting points in the general way as described in Section 5.1 can, however, not be used here.
x y z	Are given as real numbers where x, y and z are the cartesian coordinates of the point.
<	Enter into the *update-coordinates* mode.
>	Conclude the *update-coordinates* mode.
update-coordinates	See explanation under the DEFINE POINT command.

NOTES:

If the new position of a point coincides with any other point then the change is not performed. The position coincides if the distance is less than the coordinate tolerance given by the SET TOLERANCE COORDINATE command.

EXAMPLES:

The following command changes the coordinates of point P1 to (1,2,3).

```
CHANGE POINT P1 1 2 3
```

The following command adds 4.25 to the Z-coordinate of all points starting with AP21.

```
CHANGE POINT AP21* < DZ 4.25 >
```

CHANGE PROPERTY LOAD load-case TO-MASS

...	PROPERTY	LOAD	load-case	TO-MASS	load-type	grav-lc	select-geometry

PURPOSE:

The command converts certain load types to nodal masses. These load types are:

- CONCENTRATED, i.e. loads applied to nodes

Plus the following load types applied to two node beam elements:

- LINE-LOAD
- PART-LINE-LOAD
- BEAM-CONCENTRATED-LOAD

Note: Only loads applied to the two node beam element may be converted.

The conversion of loads to nodal masses is done by computing the consistent nodal forces for the loads in question and then dividing these nodal forces by the acceleration of gravity. The acceleration of gravity is taken from a previously defined gravity loadcase. The computed masses are added to any previously defined masses for the three translational components of the mass matrix. The rotational components and off-diagonal terms of the mass matrix are not changed (any previously defined rotational mass components are maintained).

However, this change of load to mass is only made on the condition that the forces to be changed and the acceleration of gravity referred to are *parallel* and with the *same sign* (the X-, Y- and Z-components are considered in combination and not individually). A load including moments is not changed, even if its translational part (the force) is parallel with the acceleration of gravity. Furthermore, a complex load (a load including an imaginary part) will not be changed (this may first be changed to a real load by removing the imaginary part and thereafter be changed to mass).

The presence of other types of load in the same load case does not prevent the LINE-LOAD, PART-LINE-LOAD and BEAM-CONCENTRATED-LOAD to be changed to mass as long as they meet the requirements above.

Loads changed to masses are not removed, rather they are marked as ‘LOAD TO BE CONVERTED TO MASS’ (this will be seen when printing the loads). Therefore, if the mesh is deleted and re-created the loads are converted to masses for the new mesh.

PARAMETERS:

load-case — The load case to be converted to mass.

load-type — The load type of load-case to be converted to mass. Relevant alternatives are: ALL-LOAD-TYPES, CONCENTRATED, LINE-LOAD, PART-LINE-LOAD and BEAM-CONCENTRATED-LOAD.

grav-lc	A previously defined load case containing the acceleration of gravity used in the computation of masses.
select-geometry	Points or lines for which loads shall be converted. See Section 5.1 on how to perform a selection.

CHECK

CHECK	CLUSTERED-NODES	...
	CLUSTERED-POINTS	
	ELEMENT-SHAPE	
	MESH-TOPOLOGY	
	NON-REGULAR-NODES	

PURPOSE:

The command checks the mesh.

The command also checks whether there are geometry points positioned closely together.

PARAMETERS:

CLUSTERED-NODES	Check whether nodes are positioned closely together and list such nodes.
CLUSTERED-POINTS	Check whether geometry points are positioned closely together and store such points in a named set.
ELEMENT-SHAPE	Check the shape of 2-D and 3-D elements.
MESH-TOPOLOGY	Check whether it is possible to create an element mesh with the given data (consistency of data).
NON-REGULAR-NODES	Identify surfaces with potential for mesh improvements by searching for so-called non-regular-nodes.

CHECK CLUSTERED-NODES

...	CLUSTERED-NODES	select-geometry	tolerance

PURPOSE:

The command detects pairs of nodes positioned closely together and prints these pairs in the message window.

Clustered nodes may be caused by unintentionally coincident geometry. In such case delete the mesh, correct the geometry and re-create the mesh.

Clustered nodes may also be caused by a defect in the mesh. In such case the commands DELETE MESH ALL and MESH ALL may resolve the matter.

PARAMETERS:

select-geometry	Select geometry. See Section 5.1 on how to perform a selection.
tolerance	The cluster tolerance, i.e. a pair of nodes closer than this distance is listed.

CHECK CLUSTERED-POINTS

...	CLUSTERED-POINTS	select-points	set-of-clustered-points	tolerance

PURPOSE:

The command picks out geometry points positioned closely together and stores these in a named set. The set of clustered points may then be printed or displayed.

PARAMETERS:

select-points	Select points. See Section 5.1 on how to perform a selection.
set-of-clustered-points	Name of a new set (to be created by this command).
tolerance	The cluster tolerance, i.e. if points are more closely positioned than this distance then they are put into the set.

CHECK ELEMENT-SHAPE

...	ELEMENT-SHAPE	select-elements	set-of-failed-elements	...

...	ANGLE	MINIMUM	min	
		MAXIMUM	max	
		BOTH-MINIMUM-AND-MAXIMUM	min	max
	ASPECT-RATIO	ratio		
	REPORT-FAILING-ELEMENTS			
	TWIST	twist-angle		
	END			

PURPOSE:

The command picks out elements failing specified criteria and stores them in the named set. The selected elements are checked successively for all specified criteria and stored in the same set until END is given. The set of failing elements may be printed or displayed.

PARAMETERS:

select-elements	Select elements to be checked. See Section 5.2 on how to perform a selection.
set-of-failed-elements	Name of a new set (to be created within this command).
ANGLE	Check the element corner angle. 1-D elements (beam, truss, spring, etc.) and contact elements cannot be checked.
MINIMUM	Check for small angles.
MAXIMUM	Check for large angles.
BOTH-MINIMUM-AND-MAXIMUM	Check for both small and large angles.
min	Smallest acceptable value of small angles.
max	Largest acceptable value of large angles.
ASPECT-RATIO	Check the element aspect ratio. The aspect ratio of an element is the longest distance between two nodes of the element divided by the shortest distance between two nodes. Only corner nodes of the elements are considered.
ratio	The largest acceptable value of the aspect ratio.

REPORT-FAILING-ELEMENTS	The elements failing will be listed on the screen in addition to being stored in the set.
TWIST	Check the element twist in terms of twist or rotation of its edges. Only 2-D elements, i.e. shell, membrane and axisymmetric elements are checked. The twist of an edge of an element is the angle by which the edge is twisted. It is calculated as explained by Figure 5.16.
twist-angle	The maximum twist angle allowed.

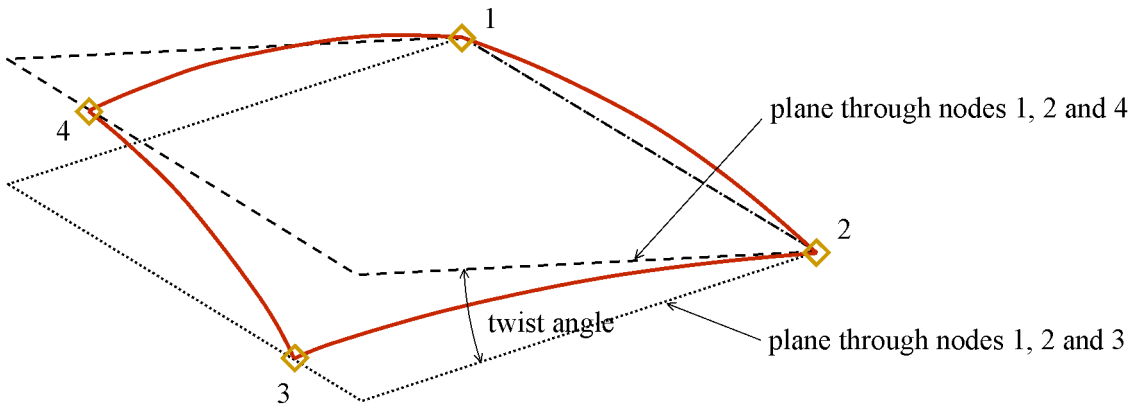


Figure 5.16 Twist of element

CHECK MESH-TOPOLOGY

...	MESH-TOPOLOGY	select-geometry

PURPOSE:

The command performs a check on whether it is possible to create a mesh for the geometry. The mesh is not created.

Creating large and complex meshes may be time-consuming. This command allows checking the consistency of the data determining the mesh without the risk of wasting time.

Geometries that cannot be meshed will be reported allowing the user to adjust the data determining the mesh. See Section 2.7.3 for constraints on the mesh.

PARAMETERS:

select-geometry — Surfaces and bodies for which a check shall be performed. See Section 5.1 on how to perform a selection.

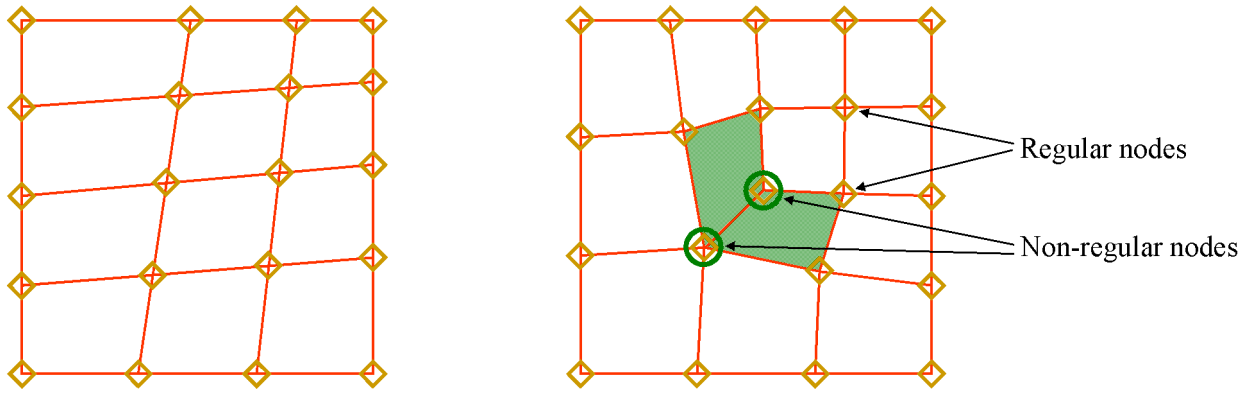
CHECK NON-REGULAR-NODES

...	NON-REGULAR-NODES	select-geometry

PURPOSE:

The command identifies surfaces with potential for mesh improvements.

The command enables the user to evaluate a quality aspect of the mesh based on the concept that a topologically square (regular) mesh is generally better than a non-square one. The command considers the corner nodes of quadrilateral elements. A node is termed regular if there are 4 elements connected to it, otherwise it is termed non-regular. A mesh that is not topologically square will have one or more non-regular nodes. See Figure 5.17.



Topologically square mesh **Topologically (and geometrically) non-square mesh**

Figure 5.17 Square mesh with only regular nodes and non-square with non-regular nodes

Only elements of surfaces (shell and membrane) and only nodes in the interior of surfaces are considered. The command will write a list of surfaces where there are elements with two or more non-regular nodes. In the example of Figure 5.17 there are two elements with two non-regular nodes each.

Among the surfaces listed there may be some surfaces where a different setting of mesh corners or number of elements will give better mesh. The total number of non-regular nodes and total number of elements with two or more non-regular nodes are also written.

PARAMETERS:

select-geometry Surfaces for which a check shall be performed. See Section 5.1 on how to perform a selection.

CONNECT

CONNECT	LAYERED	layered-name	select-surfaces
	MATERIAL	material-name	select-geometry
	SECTION	section-name	select-lines

PURPOSE:

The command connects (assigns) layered element, material and cross sectional data to the relevant geometries. The material and cross sectional data must previously have been defined by the PROPERTY MATERIAL and PROPERTY SECTION commands.

In order for an anisotropic material to be connected to a surface the surface must have been assigned the appropriate element type.

The layered element data must previously have been defined by the DEFINE LAYERED command.

A new CONNECT command for the same geometry will override previous assignments.

PARAMETERS:

LAYERED	Connect layered element data.
layered-name	Previously defined layered name.
select-surfaces	Surfaces for which the layered element data shall apply. See Section 5.1 on how to perform a selection.
MATERIAL	Connect material data.
material-name	Previously defined material name.
select-geometry	Geometry to which this material shall apply. See Section 5.1 on how to perform a selection.
SECTION	Connect cross sectional data.
section-name	Previously defined cross section name.
select-lines	Lines for which the cross sectional data shall apply. See Section 5.1 on how to perform a selection.

COPY

COPY	select-geometry	mask	trnam								
			TRANSLATION-VECTOR	[	REPEAT	n-times	]	dx	dy	dz

PURPOSE:

The command copies geometry. This may either be done by means of a geometrical transformation or through a translation defined by a vector. The transformation must previously have been defined by the command DEFINE TRANSFORMATION. The former method is the more general one as the transformation matrix may contain rotations, mirroring and even scaling. The latter method is the quicker one as the data is given 'on-the-fly'.

Practical and efficient use of the COPY command will benefit by an understanding of the concept of wild-cards; see Section 3.9.3 as well as the notes below for information on this.

PARAMETERS:

select-geometry	Select geometry to be copied. Copying a body implies copying of the related surfaces, lines and points as well. Copying a surface implies copying the related lines and points, etc. See Section 5.1 on how to perform a selection.
mask	Mask name (a wild-card name) for naming the new geometry. See the notes below.
trnam	Name of the transformation defining the copying process. This transformation must previously have been defined by the DEFINE TRANSFORMATION command.
TRANSLATION-VECTOR	Copy by means of a translational vector.
REPEAT	Make several copies by repeating the translational vector. This is optional, simply skip 'REPEAT n-times' and give the vector (dx dy dz) directly if a single copy is wanted.
n-times	Number of copies.
dx dy dz	The translational vector.

NOTES:

Normally, several geometric entities are copied by a single COPY command. A system for naming the new geometric entities is therefore required. The mask name provides this system as follows:

- Wild-card characters & and * integrated into the mask name implies that the corresponding characters in the names of the new geometry are the same as those in the names of the originals.

- Characters other than & and * in the mask name replace the corresponding characters in the names of the originals.

- If a name for a new geometry constructed from the above rules is occupied then default names will be used: PO1, PO2, ... for points, LI1, LI2, ... for lines and SU1, SU2, ... for surfaces.

A geometric entity not embraced by select-geometry but forming a part of geometry that shall be copied will also be copied. For example, if line AI123 is defined based on point BP123 then copying A* will also copy BP123 even though it does not match the mask name. The mask name will apply to such 'extra' geometric entities as well, e.g. the mask name C* for copying AI123 and BP123 will result in the new names CI123 and CP123, respectively.

If there already (partly) exists geometry at the destination of a copy then (that part of) the geometry will not be copied.

When copying geometry some properties and other data are carried over to the copy and some are not. Below are listed the properties and other data carried over to the new geometry.

- For points:
  — Boundary conditions
  — Loads

- For lines:
  — Number of elements (line division)
  — Element length ratio
  — Material property
  — Section property
  — Boundary conditions
  — Loads

- For surfaces:
  — Material property
  — Thickness
  — Boundary conditions
  — Loads

- For bodies:
  — Material property
  — Boundary conditions
  — Loads

Note: By default types of element for the lines, surfaces and bodies are not copied. Copying element type may, however, be switched on by the SET DEFAULT COPY-ELEMENT-TYPE ON command.

Note: Spring and damper elements defined for points are not copied.

EXAMPLES:

The examples below are given to illustrate how names of geometry created by the COPY command are composed. The geometry names are based on the naming system of the GENERATE command as explained and exemplified in Section 3.3.7. T1, T2, etc. are previously defined transformations. (Whether the COPY command refers to a transformation or the TRANSLATION-VECTOR option is used is irrelevant for the naming of the new geometry.)

The following command copies all geometric entities starting with A. The names of the new geometric entities will be the same as those of the originals only replacing the A by a B. For example, the copy of body AB111 will be named BB111, the copy of surface AU123 will be named BU123, etc.

```
COPY A* B* T1
```

In the following command body AB123 with all its surfaces, lines and points will be copied. The names of the new geometric entities will be the same as those of the originals only replacing the A by a C.

```
COPY AB123 C* T2
```

In the following command surface AU123 with all its lines and points will be copied. The mask name implies that the copies shall have the same names as those of the originals. But since this is not allowed default names will be used for all new geometric entities.

```
COPY AU123 * T3
```

In the following command surface AU123 with all its lines and points will be copied. The names of the new geometric entities will be the same as those of the originals only replacing the last character by 5.

```
COPY AU123 &&&&5 T4
```

In the following command surface AU123 with all its lines and points will be copied. The names of the new geometric entities will be the same as those of the originals only replacing the A by a D and the last character by 6.

```
COPY AU123 D&&&6 T5
```

CREATE

CREATE	CONNECTOR[a]	...
	CRACK[b]	
	DESCRIPTION	
	MESH	
	SUPERELEMENT[c]	

a. This option is presently inactive.
b. This option is presently inactive.
c. This option is presently inactive.

PURPOSE:

The commands create data.

PARAMETERS:

DESCRIPTION	Create a description for a material name, a cross section name or a set.
MESH	Create the FE mesh.

CREATE DESCRIPTION

...	DESCRIPTION	name	text	*
			END	

PURPOSE:

The command creates a description for material names, section names and sets. These must previously have been defined by the commands PROPERTY MATERIAL, PROPERTY SECTION and DEFINE SET, respectively. Creating descriptions is solely for documentation purposes and is not required to establish a complete model.

The descriptions are reproduced in tables produced by the PRINT command. The description of sets are also stored on the Input Interface File and transferred together with the sets to the Results Interface File.

PARAMETERS:

name	Name of a previously defined section, material or set.
text	The description consisting of maximum four character strings, each of maximum 64 characters and enclosed in single quotes. If less than four strings are to be entered conclude by entering END.

EXAMPLES:

A description for the previously defined set MYSET is given by:

```
CREATE DESCRIPTION MYSET 'This set includes all 2 node beam elements.'
'No shell elements are included.'
END
```

CREATE MESH

...	MESH	name-of-geometry	*
		ADJUST	select-surfaces
		ALL	
		CRACK[a]	
		PART	select-geometry
			ALL-SPRINGS-DAMPERS-INCLUDED

a. This option is presently inactive.

PURPOSE:

The command creates the FE mesh plus damper and spring elements. The command is equivalent to the MESH command.

CUT

CUT	select-surfaces	...	tool				
			GENERAL-PLANE	point1	point2	point3	
			GRAPHICS	drag-mouse		*	
				END			
	select-lines		PREDEFINED-PLANE	[REPEAT]	XY-PLANE	dz	[*]
					XZ-PLANE	dy	
					YZ-PLANE	dx	
					[END]		

Where point1, point2 and point3 each represent:

point-name			
DEFINE-POINT	x	y	z
	<	*update-coordinates*	>

PURPOSE:

The command cuts geometry, e.g. a surface into two surfaces and a line into two lines. Only planar, cylindrical and spherical surfaces created in a certain way may be cut. See Section 3.3.4 about this.

PARAMETERS:

select-surfaces	Surfaces to be cut. See Section 5.1 on how to perform a selection. See Section 3.3.4 about which kinds of surface may be cut.
select-lines	Lines to be cut. See Section 5.1 on how to perform a selection.
tool	Name cutting tool. This may be a point, line or a shape of type plane, see Table 5.1.
GENERAL-PLANE	Use a plane defined by three points as cutting tool.
point-name	Name of an existing point.
DEFINE-POINT	Define a temporary point now.
x y z	The cartesian coordinates of the temporary point.
<	Enter into the *update-coordinates* mode.
>	Conclude the *update-coordinates* mode.
update-coordinates	See the DEFINE POINT command for an explanation of this method for determining the coordinates of a point.

GRAPHICS	Cut by graphics means.
drag-mouse	Press, hold, drag and release the mouse to make a line in the graphic display area for cutting the selected geometry.
PREDEFINED-PLANE	Use as cutting tool a plane parallel with a cartesian plane.
REPEAT	Allows specifying several cutting planes. If given conclude the cutting by END.
XY-PLANE	The cutting plane is parallel with the cartesian XY-plane.
XZ-PLANE	The cutting plane is parallel with the cartesian XZ-plane.
YZ-PLANE	The cutting plane is parallel with the cartesian YZ-plane.
dz	The cartesian Z-coordinate of the cutting plane.
dy	The cartesian Y-coordinate of the cutting plane.
dx	The cartesian X-coordinate of the cutting plane.

NOTES:

If a line is used as cutting tool the line itself is *not* cut. The line is only a tool and not a part of the geometry.

Table 5.1 Geometry to cut and tool to use

Geometry	Tool	Cutting process
line	point	The line is cut in two at the point. The sum of number of elements for the two lines will be equal to the number of elements for the original line. Element type is inherited from the original line. See the COPY command for other properties carried over from the original.
line	line	A new point is positioned at the intersection point. The line being cut is handled as described above for line cut by point. The line being the tool is not cut.
line	plane (shape)	A new point is positioned at the intersection point. The line being cut is handled as described above for line cut by point.
surface	line	New points are positioned where the line cuts the borderlines of the surface. The borderlines are cut as described above for line cut by point. A new line is created between the two new points. The surface is cut in two. Element type is inherited from the original surface. See the COPY command for other properties carried over from the original.
surface	plane (shape)	New points are positioned where the plane cuts the borderlines of the surface. The borderlines are cut as described above for line cut by point. A new line is created between the two new points. The surface being cut is handled as described above for surface cut by line.

See also the SET NAMING CUT command.

DEFINE

DEFINE	ARC	...
	BODY	
	COORDINATE-SYSTEM	
	DAMPER	
	INTERSECTION	
	LAYERED	
	LINE	
	MASS-ELEMENT	
	NODE-LINE	
	PARAMETER	
	POINT	
	PRISM	
	ROUNDED-CORNER	
	SECTOR-CORNER	
	SET	
	SHAPE	
	SPLINE	
	SPRING	
	SURFACE	
	TRANSFORMATION	

PURPOSE:

The commands define geometry, shapes (tools for defining geometry), transformations and coordinate systems, sets and certain element types (springs, dampers, etc.).

PARAMETERS:

ARC	Define the geometry entity line/curve of type arc.
BODY	Define the geometry entity body.
COORDINATE-SYSTEM	Define a coordinate system.
DAMPER	Define the element type damper.

INTERSECTION	Define the geometry entity line/curve of type intersection.
LAYERED	Define the element type layered shell element.
LINE	Define the geometry entity line/curve of type (straight) line.
MASS-ELEMENT	Define the element type mass element.
NODE-LINE	Define the geometry entity line/curve of type node-line.
PARAMETER	Define a parameter, a value that later can be used in place of a real value.
POINT	Define the geometry entity point.
PRISM	Define the geometry entity body in a simplified way.
ROUNDED-CORNER	Define a "rounded corner", i.e. round off a corner by creating an arc where two lines meet.
SECTOR-CORNER	Define a "sector-corner", i.e. cut the corner of a surface by an arc having the surface corner point as its centre.
SET	Define a set, a selection of geometry, nodes or elements.
SHAPE	Define a shape, a tool used for creating geometry and forming FE mesh.
SPLINE	Define the geometry entity line/curve of type spline.
SPRING	Define the element type spring.
SURFACE	Define the geometry entity surface.
TRANSFORMATION	Define a transformation.

DEFINE ARC

...	ARC	name	start-point	end-point	centre-point	nelm

PURPOSE:

The command defines the geometric entity arc. The angle of the arc must be less than 180 degrees. If the distance between centre-point and start-point is different from the distance between centre-point and end-point then the arc is not circular but rather an arc with linearly varying radius.

PARAMETERS:

name	User-given name of the arc. The name is used for later reference to the line, e.g. when a surface is defined.
start-point	Name of the point defining the start point of the arc.
end-point	Name of the point defining the end point of the arc.
centre-point	Name of the point being the centre of the arc.
nelm	Number of elements to be created along the arc. See also the commands SET NUMBEROF-ELEMENTS and SET MAX-ELEMENT-LENGTH.

NOTES:

The direction of the arc going from start-point to end-point has consequence for the local coordinate system of beam elements.

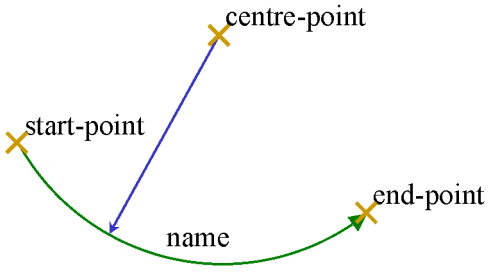


Figure 5.18 Arc

DEFINE BODY

...	BODY	name	top	bottom	side	*

PURPOSE:

The command defines the geometric entity body.

Bodies are by definition enclosed by top and bottom surfaces and by any number of side surfaces. The following requirements to a body and its mesh must be met:

- A body must logically, if not geometrically, be prismatic as shown in Figure 3.18.
- This implies that the side surfaces must be quadrilateral, if not rectangular, and extend from the top surface to the bottom surface. In the command defining bodies the side surfaces must be given in sequence.
- The top and bottom surfaces must have been defined in the same sequence starting and ending in corresponding positions. The number of borderlines of the top and bottom surfaces need not be the same but the surfaces must have mesh-corners and not-mesh-corners in corresponding positions. This means that if, say, two lines of the bottom surface correspond to one line of the top surface then there must be a not-mesh-corner in between the two lines of the bottom surface. See Figure 3.18.
- The element mesh for a body must also be prismatic, i.e. the meshes of the top and bottom surfaces must in terms of topology be equal (their corresponding borderlines/curves must have equal number of elements and they must have mesh-corner and not-mesh-corner in corresponding points). Further, the mesh of the side surfaces must be regular with no mesh refinement and with mesh-corners in the four points connecting with the top and bottom surfaces and only there.

For solid models with several bodies it is advisable to have the bodies positioned so that the surface being the top of one body is the bottom of the next body and let the bodies be located side by side. Otherwise meshing the bodies may be difficult.

PARAMETERS:

name	User-given name of the body.
top	Name of the surface defining the top of the body.
bottom	Name of the surface defining the bottom of the body.
side	Side surfaces given in sequence until the body is closed. Each surface given must adjoin the previously given side surface.

NOTES:

See also the DEFINE PRISM command.

DEFINE COORDINATE-SYSTEM

...	COORDINATE-SYSTEM	coord-name	CYLINDRICAL	start-xyz	z-axis-xyz	r-axis-xyz
			SPHERICAL	start-xyz	z-axis-xyz	r-axis-xyz

PURPOSE:

The command defines a coordinate system. Loads and boundary conditions normal to spheres and cylinders may easily be defined by referring to previously defined coordinate systems. Using a transformation combined with a cylindrical coordinate system also allows specification of loads and boundary conditions normal to a cone. Coordinates of points may be specified in cylindrical or spherical coordinate systems.

A coordinate system is identified by its type, i.e. cylindrical or spherical, and a transformation of the cartesian coordinate system of the model. It is defined by specifying its origin, defining its z-axis (cylindrical) or pole axis (spherical) and defining the ϕ=0-plane (which determines the r-axis).

PARAMETERS:

coord-name	User-given name of the coordinate system.
CYLINDRICAL	A cylindrical coordinate system is defined.
SPHERICAL	A spherical coordinate system is defined.
start-xyz	Cartesian coordinates of the origin of the coordinate system.
z-axis-xyz	Cartesian coordinates of a point defining the z- or pole axis of the coordinate system.
r-axis-xyz	Cartesian coordinates of a point defining the ϕ=0-plane (which determines the r-axis).

NOTES:

A coordinate system cannot be changed or deleted.

Coordinate systems defined are printed on the screen by the PRINT TRANSFORMATION command.

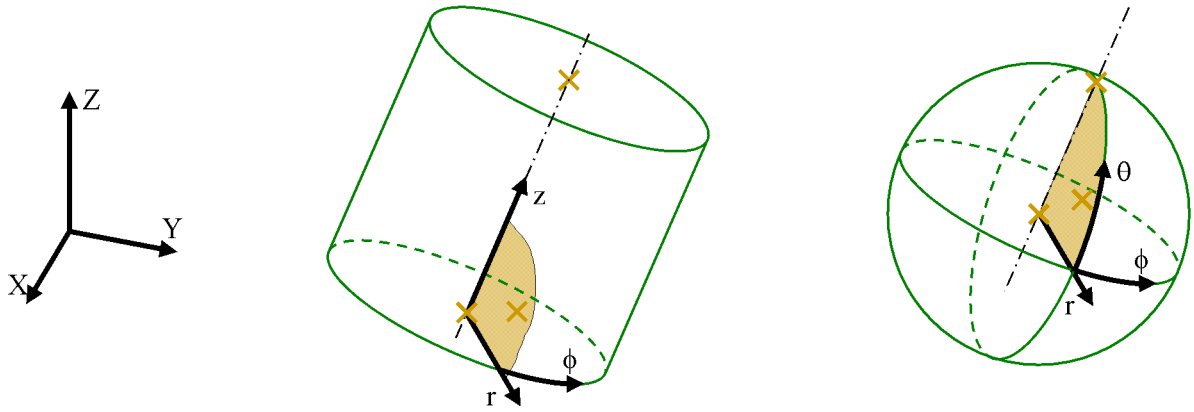


Figure 5.19 Cylindrical and spherical coordinate systems

DEFINE DAMPER

...	DAMPER	AXIAL	name	point1	point2	material-name		
		TO-GROUND	name	select-points		material-name	GLOBAL	
							TRANSFORMED	trnam

PURPOSE:

The command defines a single axial damper element between two points or several damper-to-ground elements connected to points.

The dampers are given names and their material data (damping characteristics) must previously have been defined by the PROPERTY MATERIAL command. Dampers are automatically connected to the nodes created at the points and given element numbers. The dampers are created by the MESH command.

The DELETE MESH command deletes the dampers but not their definitions meaning that when the mesh is re-created (e.g. by MESH ALL) the dampers will be re-created. The DELETE DAMPER command, however, deletes the definition of dampers.

A damper-to-ground may have from one and up to six degrees of freedom; this is specified in the PROPERTY MATERIAL command.

PARAMETERS:

AXIAL	Define an axial damper.
TO-GROUND	Define a damper-to-ground.
name	User-given name of the damper(s).
point1	Name of the point to which end 1 of the axial damper is to be connected.
point2	Name of the point to which end 2 of the axial damper is to be connected.
select-points	Select points where damper-to-ground elements are to be connected. See Section 5.1 on how to perform a selection.
material-name	Name of a previously defined material of type damper (AXIAL or TO-GROUND whichever is relevant).
GLOBAL	The material constants of the damper refer to the cartesian coordinate system of the model.
TRANSFORMED	The material constants of the damper refer to a transformed coordinate system.
trnam	Name of the transformation used.

DEFINE INTERSECTION

...	INTERSECTION	name	start-point	end-point	guiding-point	shape1	shape2	nelm

PURPOSE:

The command defines the geometric entity intersection curve. The curve is determined as the intersection between two previously defined shapes; see the DEFINE SHAPE command. Note that the intersection curve must go from one point to another defining a boundless intersection curve between two shapes is therefore not possible.

The intersection between e.g. a plane and a cylinder will give two curves one being the short way between the two points and another the long way around the cylinder. A guiding point in the vicinity of the middle of the desired intersection curve is used to select the proper one. A guiding point must be given even in cases where there is only one intersection curve (intersection between two planes).

PARAMETERS:

name	User-given name of the intersection curve.
start-point	Name of the point defining the start point of the line.
end-point	Name of the point defining the end point of the line.
guiding-point	Name of the guiding point.
shape1 shape2	Names of the two intersecting shapes. These must previously have been defined.
nelm	Number of elements to be created along the line. See also the commands SET NUMBEROF-ELEMENTS and SET MAX-ELEMENT-LENGTH.

NOTES:

The direction of the intersection going from start-point to end-point has consequence for the local coordinate system of beam elements.

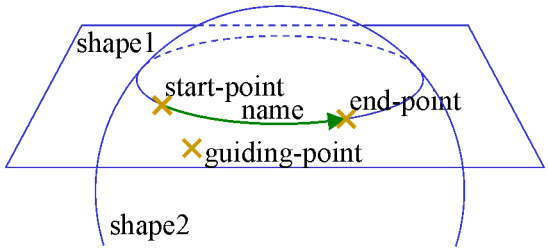


Figure 5.20 Intersection

DEFINE LAYERED

...	LAYERED	layered-name	PLATE	...
			STIFFENER	...
			END	

PLATE is followed by:

...	THICKNESS	thickness
	MATERIAL	material-name
	ECCENTRICITY	...
	SHEAR-FACTOR	sh-fact
	END	

STIFFENER is followed by:

...	SECTION	section-name	angle
	SPACING	INFINITE-PLATE	spacing
	MATERIAL	material-name	
	ECCENTRICITY	...	
	SHEAR-FACTOR	sh-fact	
	END		

ECCENTRICITY (following both PLATE and STIFFENER) is followed by:

...	ECCENTRIC	CALCULATED-NEGATIVE-Z-OFFSET			
		LOCAL-COORDINATE-OFFSET	dx	dy	dz
	NOT-ECCENTRIC				

PURPOSE:

The command defines layered element types to be used for modelling stiffened plate structures.

Stiffened plates are modelled by describing the relevant plates and stiffeners. The plates are described by their thicknesses, materials and possible eccentricities. The stiffeners are described by their cross sections, materials, positions, spacing, eccentricities and directions.

PARAMETERS:

layered-name — User-given name of the layered element. This name will be assigned to the appropriate surfaces by the CONNECT command.

PLATE	A plate layer is to be defined.
STIFFENER	A stiffener layer is to be defined.
THICKNESS	Thickness of a plate layer is to be specified.
thickness	The constant thickness of the plate. A function cannot be used.
MATERIAL	Material of the plate/stiffener is to be specified.
material-name	A previously defined material name. The material must be isotropic and elastic.
SHEAR-FACTOR	A shear factor is to be specified for the plate/stiffener layer.
sh-fact	The shear factor.
SECTION	Cross section is to be specified for the stiffeners.
section-name	A previously defined section name. The section must be of the bar type.
angle	The angle between the stiffener direction and the local x-axis of the plate element. The local x-axis is determined by the PROPERTY LOCAL-COORDINATE-SURFACE command.
SPACING	The spacing between the stiffeners is to be specified.
INFINITE-PLATE	Implies that the stiffeners are distributed in an 'infinite' plate as opposed to specifically positioned within a finite plate.
spacing	The distance between the stiffeners.
ECCENTRICITY	Whether the plate/stiffener layer is eccentric or not is to be specified.
ECCENTRIC	The plate/stiffener layer is eccentric.
CALCULATED-NEGATIVE-Z-OFFSET	The eccentricity of the plate/stiffener layer is calculated by the program so that the layer is moved in negative local z-direction (of the surface) in order not to overlap with the previous layer.
LOCAL-COORDINATE-OFFSET	The eccentricity is to be given by the user.
dx dy dz	The eccentricity given as a vector from the node to the layer. *The dx and dy components are irrelevant and should be given as 0.0.*
NOT-ECCENTRIC	The plate/stiffener layer is not eccentric.

NOTES:

A stiffened plate has different area in a section normal to the direction of the stiffeners (assuming all stiffener layers are parallel) compared with a section parallel with the stiffeners. A section parallel with the stiffeners have the bending stiffness of the plate alone whereas a section normal to the stiffener direction has a much larger bending stiffness. See Figure 5.21 for an illustration of a typical layered element consisting of a single plate layer and two stiffener layers.

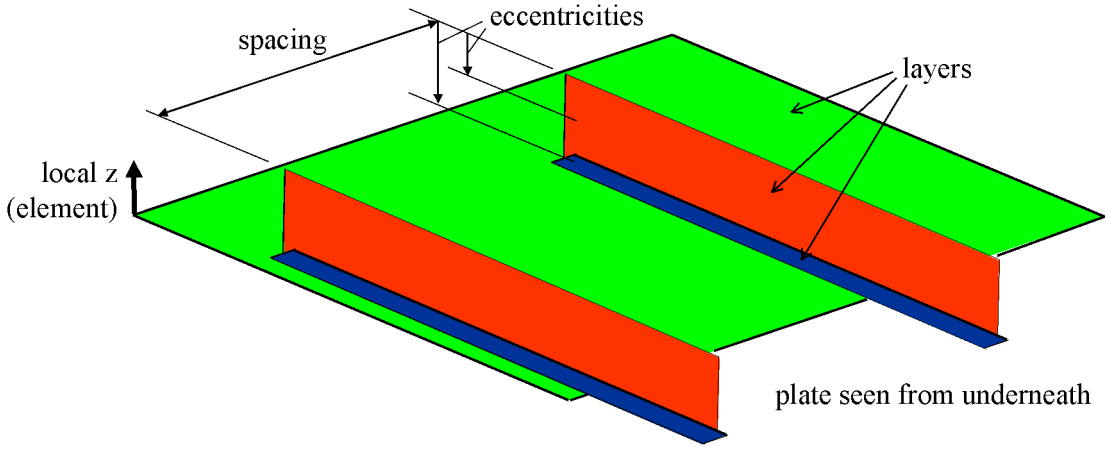


Figure 5.21 A layered element consisting of a plate layer and two stiffener layers

A stiffened plate consists of plate and stiffener layers. The stiffeners have section types, materials, eccentricities, directions and spacing. The plates have thicknesses, materials and eccentricities. The command for creating stiffened plates also covers other plate type structures like e.g. corrugated plates, reinforced concrete and lay-ups of laminates.

The command for defining the layered element refers to materials and sections. These must previously have been created. The section must be of the BAR (trapezoid) type.

One layer is defined at a time. The layers are either PLATE layers or STIFFENER layers. There is no restriction on the order or number of PLATE and STIFFENER layers. The number of layers constituting the layered element is a result of the number of times a plate/stiffener layer is defined.

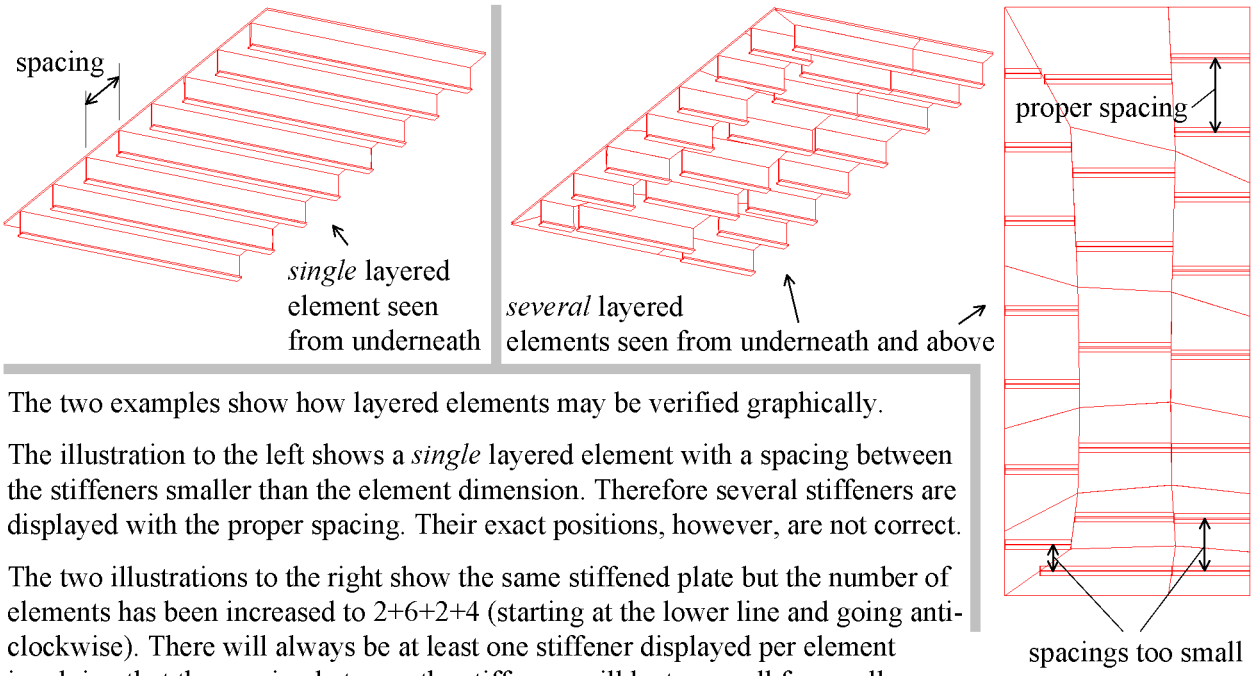
The stiffened plates require a local coordinate system. This may for convenience be selected so that the main stiffener has an angle of 0° with the local x-direction of the plate elements. The local coordinate system for the plate elements is defined by the PROPERTY LOCAL-COORDINATE-SURFACE command.

The defined layered elements are assigned to surfaces by the CONNECT command. The relevant surfaces also have to be assigned layered element type by the SET ELEMENT-TYPE SURFACE command.

A layered element may be verified graphically. See the SET GRAPHICS PRESENTATION LAYERED-ELEMENT command which offers several options for displaying the plate thickness and stiffener section. Bear in mind, however, that the stiffeners are modelled with proper spacing but not in their correct positions. The display should be interpreted with this fact in mind. The stiffeners of a layered element are displayed as follows:

- If the stiffener spacing is greater than the dimension of the element in the spacing direction then one stiffener will be displayed in the centre of the element.
- If the stiffener spacing is smaller than the element then the proper amount of stiffeners is displayed with proper spacing. The stiffener group will be centrally located on the element.

Figure 5.22 contains examples of layered element displays. The SET GRAPHICS PRESENTATION LAYERED-ELEMENT SOLID-SECTION command has been used.


spacing
single layered element seen from underneath
several layered elements seen from underneath and above
proper spacing
spacings too small

The two examples show how layered elements may be verified graphically.

The illustration to the left shows a *single* layered element with a spacing between the stiffeners smaller than the element dimension. Therefore several stiffeners are displayed with the proper spacing. Their exact positions, however, are not correct.

The two illustrations to the right show the same stiffened plate but the number of elements has been increased to 2+6+2+4 (starting at the lower line and going anti-clockwise). There will always be at least one stiffener displayed per element involving that the spacing between the stiffeners will be too small for small elements. Also, the display tend to give a somewhat confusing impression.

Figure 5.22 Display of layered elements for verification purposes

It should be noted that the results produced for layered elements by the structural analysis program Sestra consist of layers with stress components for each. I.e. there will be stress components for an idealized orthotropic layer rather than forces and moments for the stiffeners. Each layer will have stress components as for an eight node shell element. The postprocessor Xtract offers these stress components through a selection of layer and for each layer two surfaces, the upper and lower. See Xtract for more information on this.

DEFINE LINE

...	LINE	name	start-point	end-point	nelm

PURPOSE:

The command defines the geometric entity line.

Multiple line definition is possible by using wild-card names. Wild-card names may be used for the name of the line and the start and end points. For defining lines an extra wild-card feature is available through use of the plus and minus signs; this is explained in Section 3.9.3.

PARAMETERS:

name	User-given name of the line.
start-point	Name of the point defining the start point of the line.
end-point	Name of the point defining the end point of the line.
nelm	Number of elements to be created along the line. See also the commands SET NUMBEROF-ELEMENTS and SET MAX-ELEMENT-LENGTH.

NOTES:

The direction of the line going from start-point to end-point has consequence for the local coordinate system of beam elements.

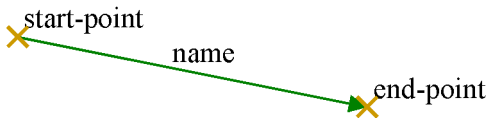


Figure 5.23 Line

DEFINE MASS-ELEMENT

...	MASS-ELEMENT	ONE-NODED	mass-element-name	select-points	material-name	...

...	GLOBAL		...	AT-POINT				
	TRANSFORMED	trnam		ECCENTRIC	x	y	z	

PURPOSE:

The command defines general one node mass elements. Mass elements are connected to points.

The mass elements are given names and their material data (masses) must previously have been defined by the PROPERTY MATERIAL command. Mass elements are automatically connected to the nodes created at the points and given element numbers. The mass elements are created by the MESH command.

The DELETE MESH command deletes the mass elements but not their definitions meaning that when the mesh is re-created (e.g. by MESH ALL) the mass elements will be re-created. The DELETE MASS-ELEMENT command, however, deletes the definition of mass elements.

A mass element may have from one and up to six degrees of freedom; this is specified in the PROPERTY MATERIAL command.

PARAMETERS:

ONE-NODED	Define a one node mass element.
mass-element-name	User-given name of the mass element.
select-points	Select points where one node mass elements are to be connected. See Section 5.1 on how to perform a selection.
material-name	Name of a previously defined material of type mass.
GLOBAL	The masses given in the PROPERTY MATERIAL command refer to the cartesian coordinate system of the model.
TRANSFORMED	The masses refer to a transformed coordinate system.
trnam	Name of the transformation used.
AT-POINT	The mass is positioned at the point.
ECCENTRIC	The mass is given an eccentric position with respect to the point.
x y z	Eccentricity vector from the point to the mass.

DEFINE NODE-LINE

...	NODE-LINE	name	start-point	next-point	*
				END	

PURPOSE:

The command defines the geometric entity node-line. This is a line where a node is to be created at every point given and no nodes in between these points. The command is used when the position of all (boundary) nodes, including mid-side nodes for higher order elements, is to be explicitly positioned. (The actual nodes are defined by using the MESH command.)

PARAMETERS:

name	User-given name of the node-line.
start-point	Name of the point defining the start point of the node-line.
next-point	Name of the points defining the node-line. When the last point has been entered conclude by END.

NOTES:

The SET NUMBEROF-ELEMENTS and SET MAX-ELEMENT-LENGTH commands cannot be used for a node-line.

As explained above the node-line specifies the position of all boundary nodes including the mid-side nodes for higher order elements. The effect of this is that the number of elements created along the node-line will depend on whether lower or higher order elements are created. For example, only two 3 node beams will be created for the node line of Figure 5.24 whereas four 2 node beams will be created. This is different from what is the case for lines and curves for which the number of elements is given explicitly.

Also note that if higher order elements are to be created for a node-line the number of point (=nodes) on the node-line must be an odd number.

The direction of the node-line going from start-point to end-point has consequence for the local coordinate system of beam elements.

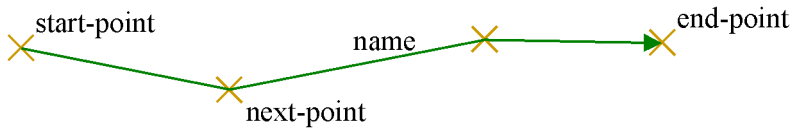


Figure 5.24 Node Line

DEFINE PARAMETER

...	PARAMETER	name	value

PURPOSE:

The command defines a value that later can be used in place of a real value in certain commands. This opens for parametric modelling to some extent.

The parameter may be used in the commands:

```
DEFINE POINT point < X para1 Y para2 Z para3 >
CHANGE POINT point < X para1 DY para2 >
```

PARAMETERS:

name User-given name of the parameter.

value The value of the parameter. May also be given as an expression, see Section 3.5.10.

NOTES:

Note that the command:

```
DEFINE POINT point parameter
```

is not allowed. This because the parameter may only be used in situations where a real value is the only alternative available (and not for example END).

DEFINE POINT

...	POINT	name	x	y	z
			<	*update-coordinates*	>

PURPOSE:

The command defines the geometric entity point. The coordinates are specified either explicitly by giving x, y and z in the cartesian coordinate system of the model or by an *update-coordinates* mode. This mode is entered into and concluded by the arrow bracket (less than and greater than) commands '<' and '>'. This mode is explained in detail on the following pages.

PARAMETERS:

name	User-given name of the point.
x y z	The cartesian coordinates of the point.
<	Enter into the *update-coordinates* mode.
>	Conclude the *update-coordinates* mode.
update-coordinates	See following pages.

NOTES:

Two points cannot have the same coordinates or more precisely: they cannot have the same coordinates within the given coordinate tolerance. See the SET TOLERANCE command.

DEFINE POINT name < *update-coordinates* >

...	<	DX	dx				>
		DY	dy				
		DZ	dz				
		X	x				
		Y	y				
		Z	z				
		point-name					
		MOVE-BY-TRANSFORMATION		trnam			
		POINT-INTERPOLATION		point1	point2	factor	
		SHAPE-INTERSECTION		shape1	shape2	shape3	
		USE-LOCAL-COORDINATE-SYSTEM		coord-name			
				GLOBAL			

When a shift from the cartesian coordinate system of the model (which is used by default) to cylindrical or spherical coordinate systems is done by the USE-LOCAL-COORDINATE-SYSTEM command then the set of commands DX, DY, DZ, X, Y, Z are substituted by either of the sets:

- DR, DPHI, DZ, R, PHI, Z
- DR, DPHI, DTHETA, R, PHI, THETA

PURPOSE:

The command defines the geometric entity point by updating or modifying a set of current coordinates. The current coordinates are, whenever updated, given in the message window on PC and in the line-mode window on Unix. The initial current coordinates (after giving the < command) will normally be those of the last point defined. The coordinates may repetitively be updated until the *update-coordinates* mode is concluded by the > command.

The *update-coordinates* mode allows use of previously defined parameters. See the DEFINE PARAMETER command.

PARAMETERS:

<	Enter into the *update-coordinates* mode.
>	Conclude the *update-coordinates* mode.
DX DY DZ	Update cartesian coordinates by adding to the current values. (Correspondingly for cylindrical and spherical coordinate systems).

dx dy dz	The additions to the current values. Previously defined parameters may be used here.
X Y Z	Update cartesian coordinates by specifying new values. (Correspondingly for cylindrical and spherical coordinate systems).
x y z	The new values. Previously defined parameters may be used here.
point-name	Update current coordinates to those of the previously defined point with name point-name.
MOVE-BY-TRANSFORMATION	Update current coordinates by moving according to the given transformation.
trnam	A previously defined transformation.
POINT-INTERPOLATION	Update current coordinates to the interpolation point between two points. An interpolation factor defines the new current coordinates as a point on the line between point1 and point2. If the factor is equal to 0 the current coordinates will be at point1; if equal to 1 the current coordinates will be at point2. Values outside the range 0 to 1 will imply extrapolation.
point1	Name of a previously defined point.
point2	Name of a previously defined point.
factor	The interpolation factor.
SHAPE-INTERSECTION	Update current coordinates to the intersection point between three shapes. The current coordinates are used to select one of the possibly two or more intersection points. Therefore, the current coordinates should be in the vicinity of the desired intersection point prior to giving the SHAPE-INTERSECTION command. See Figure 5.25.
shape1	Name of a previously defined shape.
shape2	Name of a previously defined shape.
shape3	Name of a previously defined shape.
USE-LOCAL-COORDINATE-SYSTEM	Shift to a named cylindrical or spherical coordinate system or back to the cartesian coordinate system of the model (GLOBAL).
coord-name	Name of a previously defined coordinate system.
GLOBAL	Use the cartesian coordinate system of the model.

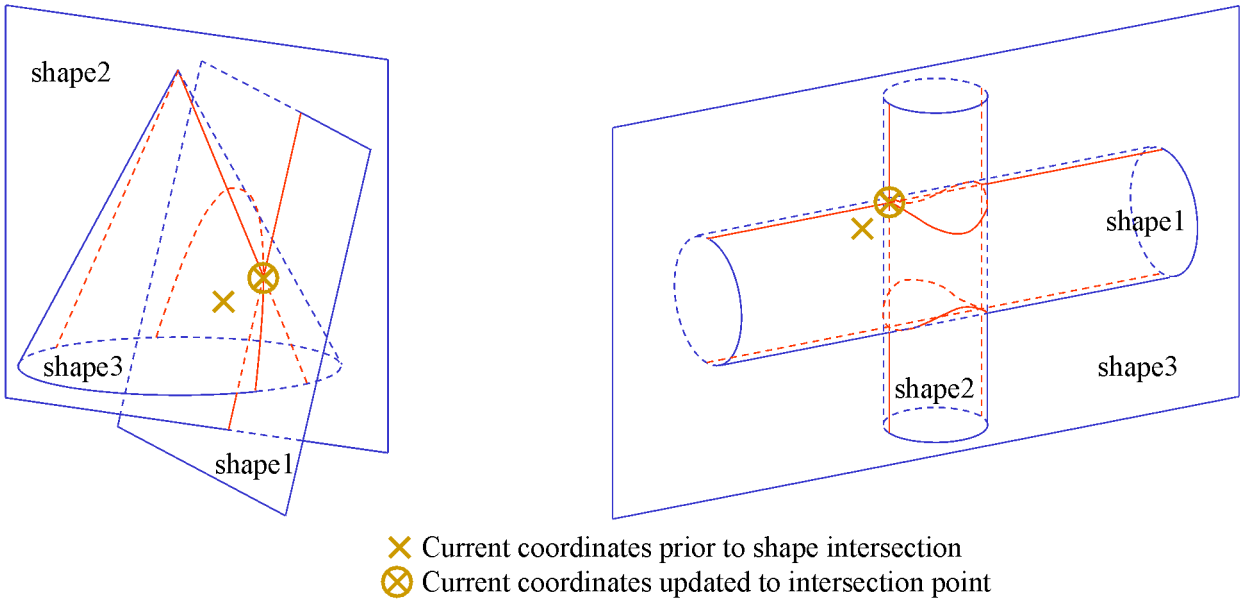


Figure 5.25 Defining point by SHAPE-INTERSECTION

DEFINE PRISM

...	PRISM	name	top	[	start-top	2nd-top END	]	bottom	start-bot END	2nd-bot END	nelm

PURPOSE:

The command defines the geometric entity body and is a short version of the DEFINE BODY command. Only the top and bottom surfaces must previously have been defined. The DEFINE PRISM command will define:

- Straight lines between corresponding points of the top and bottom surfaces
- All side surfaces of the body
- The body

The command takes advantage of how bodies are by definition: They are logically prisms enclosed by top and bottom surfaces and any number of quadrilateral side surfaces. See the DEFINE BODY command for more information on requirements to a body and its mesh.

Compared with the DEFINE BODY command some requirements must be met in order to use the DEFINE PRISM command:

- The top and bottom surfaces must be comprised of the same number of lines/curves.
- If the top and bottom surfaces are defined:
  — Starting in not prismatically corresponding points, and/or
  — In opposite directions (their surface normals points in opposite directions),
  then this can be remedied by referring to start points and second points of both surfaces (the start and second points overrule the surface definitions). In this way twisted prisms are avoided. See Figure 5.26 for an illustration of this. The points can be omitted if the definition of the top and bottom surfaces does correspond.

PARAMETERS:

name	User-given name of the body.
top	Name of the surface defining the top of the body.
bottom	Name of the surface defining the bottom of the body.
start-top	Start point of the top surface corresponding to a start point of the bottom surface.
2nd-top	Second point of the top surface which together with the start point defines the rotational direction of the top surface.

start-bot — Start point of the bottom surface corresponding to a start point of the top surface.

2nd-bot — Second point of the bottom surface which together with the start point defines the rotational direction of the bottom surface.

nelm — Number of elements to be created along the automatically generated straight lines between the top and bottom surfaces. See also the commands SET NUMBEROF-ELEMENTS and SET MAX-ELEMENT-LENGTH.

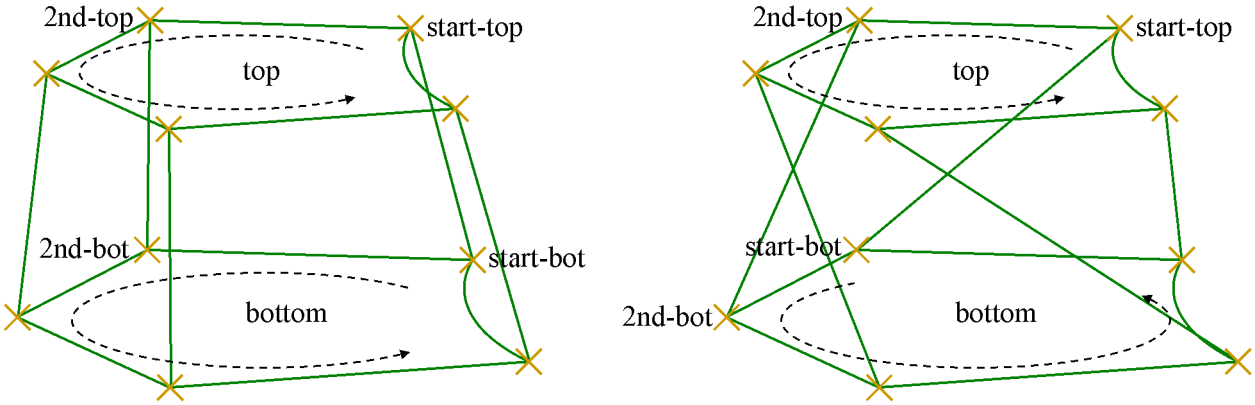


Figure 5.26 Defining a body by DEFINE PRISM — the effect of start and second points

DEFINE ROUNDED-CORNER

...	ROUNDED-CORNER	line1	line2	radius	CORNER
					CORNER-COMPLEMENT
					NONE

PURPOSE:

The command rounds off a corner by creating an arc where two lines meet. The arc will be tangential to both lines. If the lines are borderlines of a surface then the surface is divided into two surfaces. Either or none of these surfaces may be deleted. See Figure 5.27.

In case there is no surface the delete options decide which part of the corner lines to be deleted.

Note: The two lines must be straight and meet in a point.

Note: The program proposes a value for the radius being the length of the shortest of the two lines.

Note: The lines (and surface) cannot have mesh.

PARAMETERS:

line1	The first line.
line2	The second line.
radius	The radius of the arc to create.
CORNER	Delete the CORNER surface.
CORNER-COMPLEMENT	Delete the CORNER-COMPLEMENT surface.
NONE	Delete none of the surface.

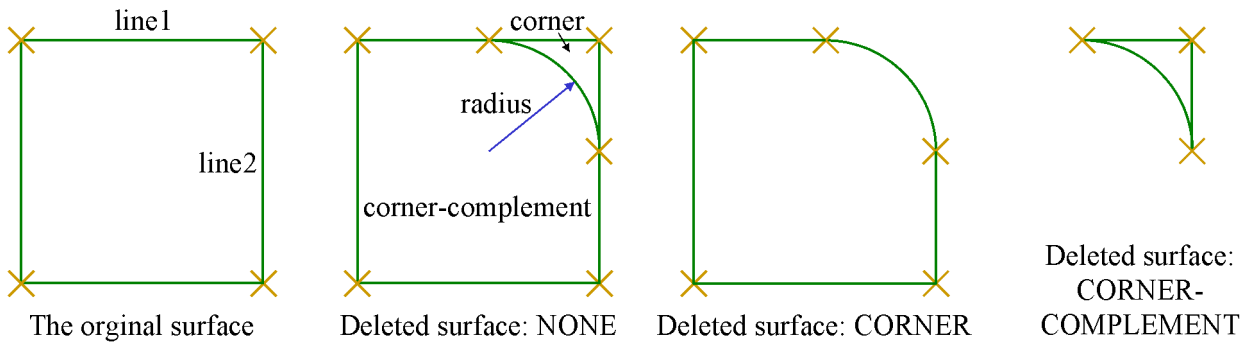


Figure 5.27 Round off a corner by DEFINE ROUNDED-CORNER

DEFINE SECTOR-CORNER

...	SECTOR-CORNER	radius			centre-pnt	surface	CORNER
							CORNER-COMPLEMENT
		DISTANCE	pnt1	pnt2			NONE
						END	

PURPOSE:

The command creates an arc with the specified radius and centre in the selected point. The arc cuts the selected surface and its borderlines. The arc cuts the surface into two surfaces. Either or none of these surfaces may be deleted.

Alternatively to giving the radius explicitly it may be given as the distance between two arbitrary points.

Having selected which surface to delete (or none) you can exit from the command (END) or continue selecting more surfaces to cut. These additional surfaces must be connected to the same point and the same radius is used for the arcs (the radius is specified only once for all surfaces). This functionality makes it easy to create a circular hole where co-planar surfaces meet in a point. See Figure 5.28.

Note: The lines cut by the arc must be straight and connect to the centre point of the arc. See Figure 5.29.

Note: The surface to be modified cannot have mesh.

PARAMETERS:

radius	The radius of the arc to create.
DISTANCE	The radius is the distance between two points.
pnt1	The first of two points defining the radius.
pnt2	The second of two points defining the radius.
centre-pnt	Centre of the arc to create.
surface	Surface to cut by the arc, it must be connected to the point centre-pnt.
CORNER	Delete the CORNER surface.
CORNER-COMPLEMENT	Delete the CORNER-COMPLEMENT surface.
NONE	Delete none of the surface.

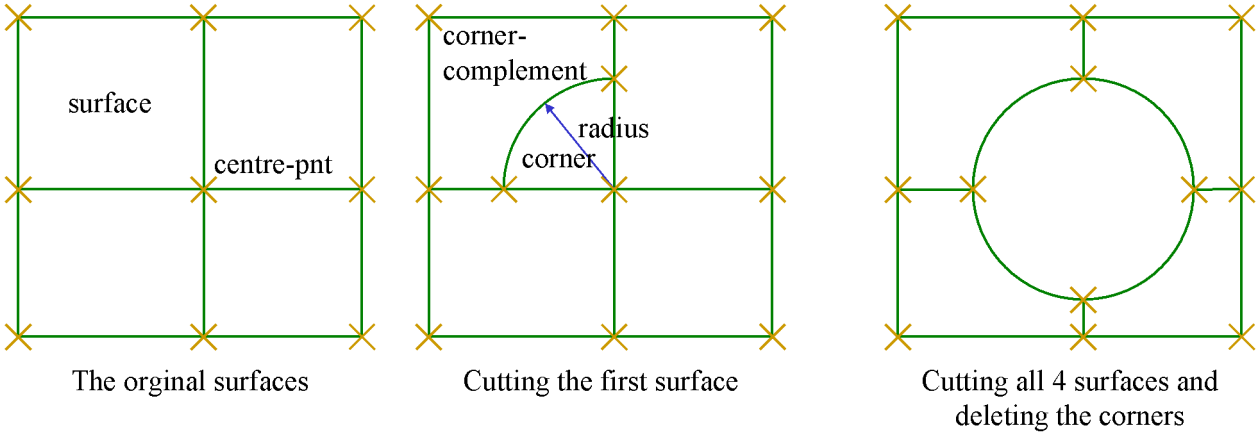


Figure 5.28 Create a hole by DEFINE SECTOR-CORNER

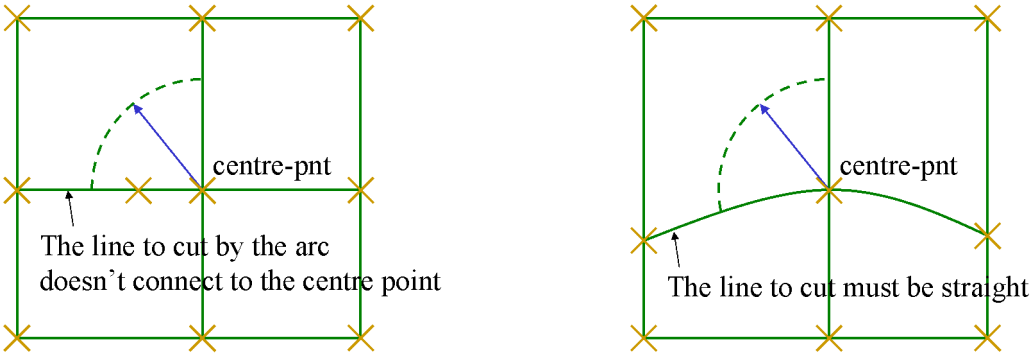


Figure 5.29 Examples where DEFINE SECTOR-CORNER cannot be used

DEFINE SET

...	SET	name	{	INTERSECTION-WITH	...	set-name		*	}*
						ALL			
						BODIES	select-bodies		
				SUBTRACT-BY		ELEMENTS	select-elements		
						LINES	select-lines		
						NODES	select-nodes		
				UNION-WITH		POINTS	select-points		
						SPRING-DAMPER	select-spring-damp		
						SURFACES	select-surfaces		
						END			
			END						

PURPOSE:

The command defines a set — a named selection — of geometry, elements or nodes.

A set may be referred to in commands where selecting geometry, elements or nodes is required.

The DEFINE SET command defines a new set while the CHANGE SET command changes an existing set. The command syntaxes of these two commands are identical and based on standard set operators.

Initially, i.e. after giving the command DEFINE SET and entering a name of the set, the set is empty. The first operation to do will therefore be to add to the set by selecting the UNION-WITH command. Thereafter, repetitive set operations may be performed until the content of the set is as desired. The operations are executed consecutively, the order of the operations is therefore of consequence.

When either of the three set operators — UNION-WITH, SUBTRACT-BY, INTERSECTION-WITH — are chosen, repetitive selections of geometry, elements and nodes may be performed. Conclude this sequence by entering END. Then a new operator may be chosen and new repetitive selections may be performed. Finally, the definition (or changing) of the set is concluded by entering another END (rather than one of the set operators).

PARAMETERS:

name — User-given name of the set to define, maximum 8 characters.

INTERSECTION-WITH — All geometry, elements and nodes except for those subsequently selected will be removed from the set. I.e. the new contents of the set will be the intersection between the current contents of the set and the subsequent selection.

SUBTRACT-BY	The subsequently selected geometry, elements and nodes will be removed from the set.
UNION-WITH	The subsequently selected geometry, elements and nodes will be added to the set.
set-name	A previously defined set.
ALL	All points, all lines, all surfaces, all bodies, all elements and all nodes are selected.
BODIES	Bodies are to be selected.
select-bodies	Select bodies. See Section 5.1 on how to perform a selection.
ELEMENTS	Elements are to be selected.
select-elements	Select elements. See Section 5.2 on how to perform a selection.
LINES	Lines are to be selected.
select-lines	Select lines. See Section 5.1 on how to perform a selection.
NODES	Nodes are to be selected.
select-nodes	Select nodes. See Section 5.2 on how to perform a selection.
POINTS	Points are to be selected.
select-points	Select points. See Section 5.1 on how to perform a selection.
SPRING-DAMPER	Spring or damper elements are to be selected.
select-spring-damp	Select spring and/or damper elements. These are selected in a similar way as selecting nodes and elements; see Section 5.2.
SURFACES	Surfaces are to be selected.
select-surfaces	Select surfaces. See Section 5.1 on how to perform a selection.

EXAMPLES:

```
DEFINE SET SETA UNION LINE AI1* END END END END
DEFINE SET SETB UNION LINE ( AI&1 AJ* ) END END END END
DEFINE SET SETC UNION LINE SETA END INTERSECT LINE SETB END END END END
```

At this point SETC will only contain the line AI11 (if this line exists).

DEFINE SHAPE

...	SHAPE	PLANE	name	point1	point2	point3				
		SPHERE	name	centre	r					
		CYLINDER	name	point1	point2	r				
		CONE	name	point1	point2	r1	r2			
		INTERPOLATION	name	point1	line1	*	point2	line2	*	
					END			END		

PURPOSE:

The command defines a shape. Shapes are tools in the form of surfaces and are used for:

- Projection of nodes created during meshing onto surfaces (see the DEFINE SURFACE and SET PROJECTION commands)
- Defining geometry by determining the intersection line between two shapes or intersection point between three shapes (see the DEFINE INTERSECTION and DEFINE POINT commands)

There are four basic shapes available; see Figure 5.30.

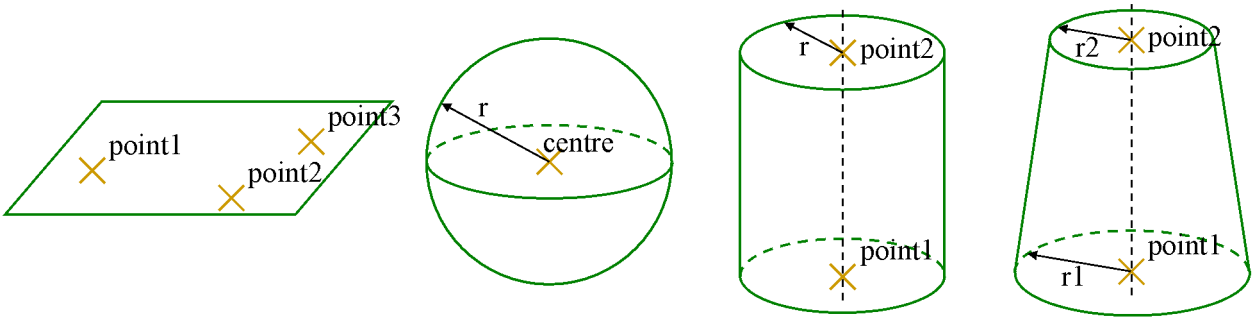


Figure 5.30 Basic shapes

Additionally, a free form shape being an interpolation between two sets of chained lines/curves is available. Each chain consists of a starting point and a sequence of lines, arcs and/or splines. A number of straight lines between the two chains form (twisted) panels. These panels constitute the shape; see Figure 5.31.

Each straight line is drawn between a pair of points on the two chains. These two points are spaced at equal distances relative to the whole lengths of the two chains. The spacings are made small enough to fulfil the following requirement: The deviation between the secant from the previous to the new point and the curve itself should not exceed a coordinate tolerance. This tolerance is set by the SET TOLERANCE COORDINATES command.

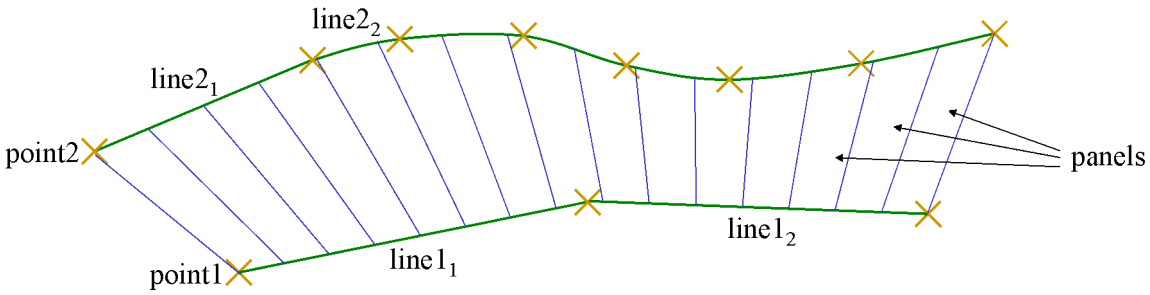


Figure 5.31 Interpolation shape

PARAMETERS:

PLANE	Define a shape being a plane.
SPHERE	Define a shape being a sphere.
CYLINDER	Define a shape being a cylinder.
CONE	Define a shape being a cone.
INTERPOLATION	Define a shape being an interpolation between two unconnected curves.
name	User-given name of the shape.
point1 point2 point3 centre	Points used for defining the shapes.
r r1 r2	Radii used for defining the shapes.
line1	Names of lines, arcs and/or splines of the first curve. Conclude entering names by END.
line2	Names of lines, arcs and/or splines of the second curve. Conclude entering names by END.

DEFINE SPLINE

...	SPLINE	name	start-point	next-point	*	nelm
				END		

PURPOSE:

The command defines the geometric entity spline. This is a curve in the form of a general B-spline of the 4th order (degree 3) interpolated between the points. There is no limit to the number of points defining the spline.

PARAMETERS:

name	User-given name of the spline.
start-point	Name of the point defining the start point of the spline.
next-point	Name of the next point of the spline. When the last point has been entered conclude by entering END.
nelm	Number of elements to be created along the spline. See also the commands SET NUMBEROF-ELEMENTS and SET MAX-ELEMENT-LENGTH. The number of elements is independent of the number of points defining the spline. This implies that nodes will not necessarily be created in the points along the spline.

NOTES:

The direction of the spline going from start-point to end-point has consequence for the local coordinate system of beam elements.

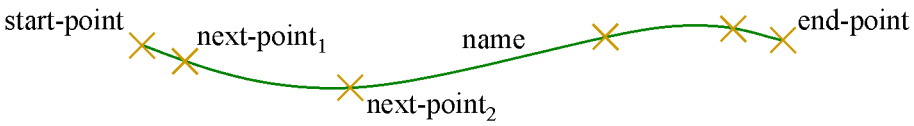


Figure 5.32 Spline

DEFINE SPRING

...	SPRING	AXIAL	name	point1	point2	material-name		
		TO-GROUND	name	select-points		material-name	GLOBAL	
							TRANSFORMED	trnam

PURPOSE:

The command defines a single axial spring element between two points or several spring-to-ground elements connected to points.

The spring elements are given names and their material data (spring stiffness) must previously have been defined by the PROPERTY MATERIAL command. Spring elements are automatically connected to the nodes created at the points and given element numbers. The spring elements are created by the MESH command.

The DELETE MESH command deletes the spring elements but not their definitions meaning that when the mesh is re-created (e.g. by MESH ALL) the spring elements will be re-created. The DELETE SPRING command, however, deletes the definition of spring elements.

A spring-to-ground may have from one and up to six degrees of freedom; this is specified in the PROPERTY MATERIAL command.

PARAMETERS:

TO-GROUND	Define a spring-to-ground element.
AXIAL	Define an axial spring element.
name	User-given name of the spring.
select-points	Select points where spring-to-ground elements are to be connected. See Section 5.1 on how to perform a selection.
point1	Name of the point to which end 1 of the axial spring is to be connected.
point2	Name of the point to which end 2 of the axial spring is to be connected.
material-name	Name of a previously defined material of type spring (AXIAL or TO-GROUND, whichever is relevant).
GLOBAL	Use the cartesian coordinate system of the model.
TRANSFORMED	The spring stiffness matrix refers to a transformed coordinate system.
trnam	Name of the transformation used.

DEFINE SURFACE

...	SURFACE	name	[shape]	first-line-name			opposite-line-name	
				...	{	[MESH-CORNERS or NOT-MESH-CORNERS]	line-name	}*
				...	{	[MESH-CORNERS or NOT-MESH-CORNERS]	point-name	}*
					END			

PURPOSE:

The command defines the geometric entity surface. A surface can have any number of borderlines/curves.

Surfaces can be defined as follows; see Figure 5.33:

a By giving the name of a line/curve and another line/curve not connected to the first one. A quadrilateral surface is formed in-between these two opposite lines/curves. Unless they already exist, two straight lines connecting the two selected lines/curves will be created. The lines created are given default names.

b By giving the names of all lines/curves enclosing the surface

c By giving the names of the points surrounding the surface. Unless they already exist, straight lines between these points will be created. The lines created are given default names. The list of points is concluded either by entering END or by closing the surface by giving the first point once more.

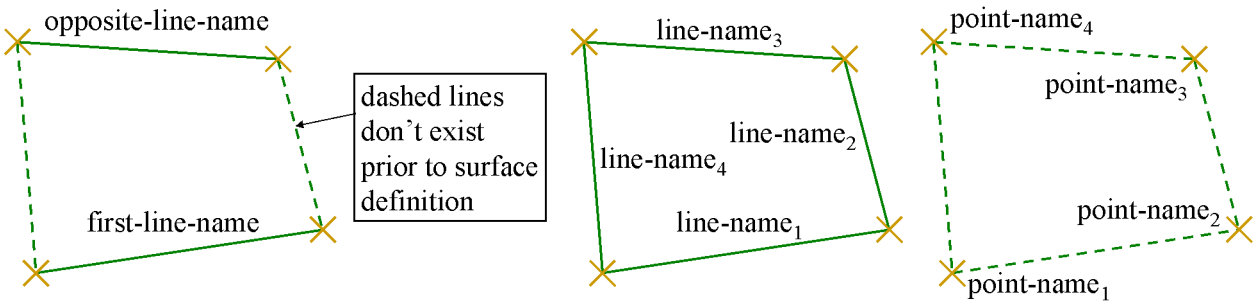


Figure 5.33 Defining surfaces

Note: Defining a surface by referring to points can only be done when *all* enclosing lines/curves are straight lines. A curve, e.g. an arc, will not be recognised by the surface definition and a straight line will be created. See Figure 5.34.

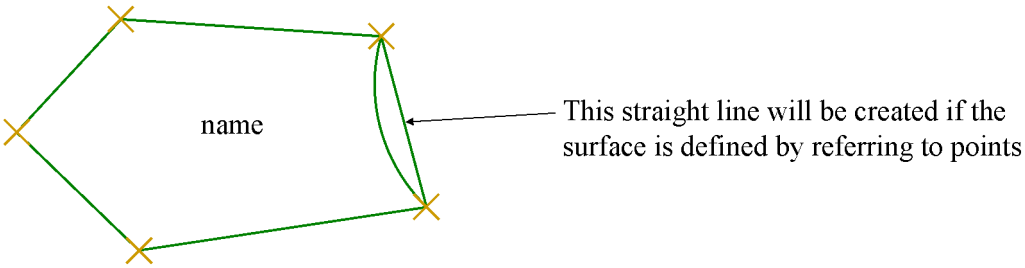


Figure 5.34 Defining a surface using lines or points

The commands MESH-CORNERS and NOT-MESH-CORNERS are optional and work as follows:

- If lines are used to define the surface then the point in between the last given line and the next one shall be a mesh-corner.
- If points are used to define the surface then the last given point shall be a mesh-corner.
- All subsequent points will be mesh-corners until the command NOT-MESH-CORNERS is given.
- Mesh-corner is the default choice. This implies that the first relevant option is NOT-MESH-CORNERS (to counteract the default). Therefore, completely omitting the MESH-CORNERS and NOT-MESH-CORNERS options involves that all points surrounding the surface will be mesh-corners.

Note: You may skip giving information on mesh-corners/not-mesh-corners when defining the surface as this may easily be defined later on by the SET MESH CORNER-TYPE command.

Note: You may even skip giving information on mesh-corners/not-mesh-corners altogether as the automatic mesh creation will optionally overrule mesh-corner settings in corner points where the angle is larger than a certain value. The default setting is such that all corners larger than 150° will be not-mesh-corner irrespective of the mesh-corner/not-mesh-corner setting.

PARAMETERS:

name	User-given name of the surface.
shape	Name of a previously defined shape onto which the nodes created for the surface will be projected. For plane surfaces this entry is omitted. For curved surfaces omission of a shape for projection may not give the desired mesh; see Section 3.4.5 on this.
first-line-name	Name of the first of two previously defined borderlines/curves spanning a surface.
opposite-line-name	Name of a previously defined borderline/curve opposite the first one.
MESH-CORNERS	Points will be mesh-corners.

NOT-MESH-CORNERS	Points will be not-mesh-corners.
line-name	Names of previously defined borderlines/curves. The lines/curves must be given consecutively so that each line/curve given adjoin the previous one. The last line/curve given must adjoin the first one.
point-name	Names of previously defined points surrounding the surface. Conclude the list either by entering END or by giving the first point once more.

NOTES:

The sequence in which the lines/curves (or points) are given when defining the surface has consequence for the local coordinate system of surface elements; see Section 3.12.3.

DEFINE TRANSFORMATION

...	TRANSFORMATION	trnam	...

...	{	IDENT					}*
		MIRROR	point1	point2			
		ROTATE	POINT-NAMES	start-point	end-point	centre	
			ANGLES	degree	start-point	end-point	
					line-name		
		TRANSLATE	DISPLACEMENTS	disx	disy	disz	
			POINT-MOVE	from-point	to-point		
		SCALE	scale-x	scale-y	scale-z		
	END						

PURPOSE:

The command defines geometrical transformations. Transformations are used in copying geometric entities, for defining boundary conditions and loads in transformed (askew) coordinate systems and for orientating spring-to-ground and damper-to-ground elements.

Transformations can be built up by using one or more of the four types of transformation: mirror, rotate, translate and scale. All transformations given for the same transformation matrix are chained in the given order. The END alternative concludes the building up of the transformation matrix. Do not abort the command by entering '..' as the definition of the transformation will then be discarded.

PARAMETERS:

trnam — User-given name of the transformation matrix.

IDENT — Reset the transformation matrix. All previous transformations are discarded.

MIRROR — Define a mirror transformation about a plane normal to the line between two points and halfway between them.

point1 point2 — Names of points used to define a mirror transformation.

ROTATE — Define a rotational transformation.

POINT-NAMES — The rotation is defined by a start point, an end point and a centre point as follows: Imagine a plane through the three points and an axis perpendicular to this plane and through the centre point. The rotation will then be the angle formed in the plane about the axis and from the start to the end point.

ANGLES — The rotation is defined by an angle about an axis. After this command an angle in degrees is given. If the entry following the angle is a point name then another point

	name is expected, the two points defining an axis. If a line name is entered then this is taken as the axis.
start-point end-point	Name of start and end points for either of the rotation methods.
centre	Centre point about which the rotation is performed.
degree	Rotation angle in degrees.
line-name	Name of a line to be used as an axis.
TRANSLATE	Define a translational transformation.
DISPLACEMENTS	Translate by giving displacements.
POINT-MOVE	Translate by moving from a point to another point.
disx disy disz	Displacements in the X-, Y- and Z-directions of the cartesian coordinate system of the model.
from-point to-point	Name of points defining the translation.
SCALE	Scale by given scaling factors in the X-, Y- and Z-directions.
scale-x scale-y scale-z	Scaling factors in the X-, Y- and Z-directions.

NOTES:

All types of transformation are relevant for copying geometry.

Only the rotation and mirroring types of transformation are relevant for defining loads.

Only the rotation type of transformation is relevant for defining a boundary condition and orientating sprint-to-ground and damper-to-ground elements.

The command PROPERTY TRANSFORMATION also defines transformations but this can only define rotational transformations.

DELETE

DELETE	CONNECTOR[a]		
	CRACK[b]		
	DAMPER	AXIAL	damper-name
		TO-GROUND	damper-name
	GEOMETRY	select-geometry	
	LAYERED	layered-name	
	MASS-ELEMENT	ONE-NODED	mass-element-name
	MESH	...	
	PROPERTY	...	
	SET	set-name	
	SHAPE	shape-name	
	SPRING	AXIAL	spring-name
		TO-GROUND	spring-name
	TRANSFORMATION	trnam	

a. This option is presently inactive.
b. This option is presently inactive.

PURPOSE:

The command deletes previously defined geometric entities, properties and other data.

PARAMETERS:

DAMPER — Delete a damper element name. Elements are not deleted, only the selected damper element name which has been defined by the DEFINE DAMPER command.

AXIAL — Delete an axial damper.

TO-GROUND — Delete a damper-to-ground.

damper-name — Name of the damper.

GEOMETRY — Delete geometric entities. See the note below.

select-geometry — Geometry to delete. See Section 5.1 on how to perform a selection.

LAYERED	Delete a layered element name. Finite elements are not deleted, only the selected layered element name which has been defined by the DEFINE LAYERED command. See the note below.
layered-name	Name of the layered element.
MASS-ELEMENT	Delete a mass element name. Elements are not deleted, only the selected mass element name which has been defined by the DEFINE MASS-ELEMENT command.
ONE-NODED	Delete a one node mass.
mass-element-name	Name of the mass.
MESH	Delete a FE mesh created. See a separate description.
PROPERTY	Delete properties. See a separate description.
SET	Delete a set.
set-name	Name of set.
SHAPE	Delete a shape defined by the DEFINE SHAPE command.
shape-name	Name of shape.
SPRING	Delete a spring element name. Elements are not deleted, only the selected spring element name which has been defined by the DEFINE SPRING command.
AXIAL	Delete an axial spring.
TO-GROUND	Delete a spring-to-ground.
spring-name	Name of the spring.
TRANSFORMATION	Delete a transformation defined by either the DEFINE TRANSFORMATION or the PROPERTY TRANSFORMATION commands. See the note below.
trnam	Name of the transformation.

NOTES:

Deleting geometry will not automatically delete an element mesh already created for that geometry. The mesh has to be deleted specifically. The fact that the mesh remains after deleting the geometry cannot be taken advantage of in any way as properties related to the geometry are lost with the deletion of the geometry.

Do not delete a layered element type connected to a surface.

DELETE TRANSFORMATION has the same effect as the DELETE PROPERTY TRANSFORMATION command.

DELETE MESH

...	MESH	ALL	
		PART	select-geometry
			select-elements
			select-nodes

PURPOSE:

The command deletes the whole or parts of the FE mesh (elements) created.

PARAMETERS:

ALL	Delete the whole mesh, i.e. all elements of all types and all nodes.
PART	Delete part of the mesh only. See the note below.
select-geometry	Identify the mesh to delete by giving the name of the relevant geometry. See Section 5.1 on how to perform a selection.
select-elements	Select elements. See Section 5.2 on how to perform a selection.
select-nodes	Select nodes. See Section 5.2 on how to perform a selection.

DELETE PROPERTY

...	PROPERTY	BOUNDARY				select-geometry	
		INITIAL-DISPLACEMENT				select-geometry	
		INITIAL-VELOCITY				select-geometry	
		LINEAR-DEPENDENCY				select-nodes	
		LOAD	{	load-case	load-type	select-geometry	}*
			END				
		LOCAL-COORDINATE-BEAM				select-lines	
		MATERIAL				material-name	
		POINT-MASS				select-points	
		SECTION				section-name	
		THICKNESS				select-surfaces	
		TRANSFORMATION				trnam	

PURPOSE:

The command deletes properties defined by the PROPERTY command.

PARAMETERS:

See the PROPERTY command for explanations of the various property types. For select-geometry, select-surfaces, select-lines, select-points and select-nodes see Section 5.1 and Section 5.2.

NOTES:

Deleting LINEAR-DEPENDENCY will change the boundary condition to FREE. An exception to this is the case where the boundary condition was defined as SUPER before the definition of the linear dependency. In such case the boundary condition will return to SUPER.

Parts of a load may be deleted by selecting only some of the load types and/or selecting only some of the geometry subjected to the load.

DELETE PROPERTY TRANSFORMATION has the same effect as the DELETE TRANSFORMATION command.

DISPLAY

DISPLAY	CONNECTOR[a]	
	BODY	select-geometry
	SURFACE	
	LINE	
	POINT	
	GEOMETRY	
	MESH	
	SHAPE	select-shapes
	ELEMENT	select-elements
	NODE	select-nodes

a. This option is presently inactive.

PURPOSE:

The command displays the whole or selected geometry, shapes, elements and nodes. See section Section 3.12.1 for examples of use of this command.

PARAMETERS:

BODY	Display selected bodies (and surfaces, lines and points).
SURFACE	Display selected surfaces (and lines and points).
LINE	Display selected lines (and points).
POINT	Display selected points.
select-geometry	Select geometry to display. See Section 5.1 on how to perform a selection.
GEOMETRY	Display the whole geometry.
MESH	Display the whole element mesh.
SHAPE	Display selected shapes with broken lines. Except for the sphere, the shapes are infinite in space. They are nevertheless displayed limited by the points defining them.
select-shapes	Shapes are selected in a similar way as geometry, see Section 5.1. Wild-card may, however, not be used for selecting shapes, their names must be given explicitly.
ELEMENT	Display selected elements.
select-elements	Select elements to display. See Section 5.2 on how to perform a selection.

NODE	Display selected nodes.
select-nodes	Select nodes to display. See Section 5.2 on how to perform a selection.

EXIT

EXIT

PURPOSE:

The command terminates the execution of Prefem.

Note: As explained in Section 2.13.1, whether or not to write the Input Interface File is normally controlled by Manager. If you want to produce the file you should check the appropriate box prior to starting Prefem. The Input Interface File is then automatically written when you exit Prefem using the command EXIT. (This makes the Prefem command WRITE superfluous.)

Note: If you in Windows close the Prefem window by the X in the upper right corner (or by the Close (Alt+F4) command of the window menu) then the Input Interface File will not be written even though you have requested this when starting Prefem. This feature may be used if you change your mind and decide not to write the file after having started Prefem.

EXTRUDE

First select geometry to copy and geometry to extrude, for each give a prefix for geometry names created:

EXTRUDE	geometry-to-copy	copy-prefix	geometry-to-extrude	extrude-prefix	...

Secondly, select the global cartesian or a specified cylindrical coordinate system for the operation:

...	GLOBAL										...
	CYLINDRICAL	orig-x	orig-y	orig-z	zaxi-x	zaxi-y	zaxi-z	raxi-x	raxi-y	raxi-z	

Thirdly, give the number of copies/extrusions to make and the vectors defining the copy/extrude process:

...	n-copy-extrude	{	[REPEAT	n-times]	dx	dy	dz	}*	...
					dphi				
		END							

Finally, choose only copying or also extruding; and if extruding, what to extrude:

...	COPY-ONLY
	POINT-TO-LINE
	LINE-TO-SURFACE
	SURFACE-TO-BODY

PURPOSE:

The command copies and extrudes a 2-D geometry to form a 3-D geometry, alternatively a 1-D geometry to form a 2-D geometry. The geometry to copy and extrude must exist. The command is organised as follows (see also the command syntax above):

- Select geometry to copy. This may for example be the surfaces of a bulkhead which shall be copied one or several times to make additional bulkheads. A prefix for the geometry names of the copies is given to enable identifying the copies later on.

- Select geometry to extrude. Points are extruded to lines, lines to surfaces and surfaces to bodies. Lines may for example be selected to be extruded to the skin and deck of a ship. When extruding lines (to surfaces) also the points connected to the lines are extruded (to lines). A prefix for the names of the extruded geometry is given to enable identifying this later on.

- Select the global cartesian or a specified cylindrical coordinate system for the copy/extrusion operation. The cylindrical coordinate system is defined by giving the cartesian coordinates of three points: The origin of the cylindrical coordinate system, a point on its z-axis and a point defining its r-axis (or phi=0 plane).

- Give the number of copies/extrusions to make. The corresponding number of vectors must be given either one by one or several at a time by the optional REPEAT n-times command. Having chosen the global cartesian coordinate system the vectors are given in terms of X-, Y- and Z-components. Having chosen a cylindrical coordinate system only the ϕ-component is given. It follows that copying/extrusion can only be done in the circumferential (ϕ) direction.

- The final input to the EXTRUDE command is to choose between the following options:
  - — Copy geometry-to-copy but do not extrude anything. The geometry-to-extrude becomes irrelevant in this case.
  - — Copy geometry-to-copy and extrude points belonging to geometry-to-extrude to lines. Even if the geometry-to-extrude includes lines and surfaces only points will be extruded.
  - — Copy geometry-to-copy and extrude points and lines belonging to geometry-to-extrude to lines and surfaces, respectively.
  - — Copy geometry-to-copy and extrude points, lines and surfaces belonging to geometry-to-extrude to lines, surfaces and bodies, respectively.

PARAMETERS:

geometry-to-copy	Select geometry to copy. See Section 5.1 on how to perform a selection. Bodies cannot be selected.
copy-prefix	Prefix for geometry names of the copies. This prefix precedes default names.
geometry-to-extrude	Select geometry to extrude. See Section 5.1 on how to perform a selection. Bodies cannot be selected.
extrude-prefix	Prefix for the names of the extruded geometry. This prefix precedes default names.
GLOBAL	Copy/extrude in the global cartesian coordinate system.
CYLINDRICAL	Copy/extrude in a cylindrical coordinate system which is subsequently defined.
orig-x orig-y orig-z	Cartesian coordinates of the origin of the cylindrical coordinate system.
zaxi-x zaxi-y zaxi-z	Cartesian coordinates of a point defining the z-axis of the cylindrical coordinate system.
raxi-x raxi-y raxi-z	Cartesian coordinates of a point defining the r-axis of the cylindrical coordinate system. The point need not be on the r-axis rather it must be positioned in the ϕ=0 plane.
n-copy-extrude	The number of copies/extrusions to make.
REPEAT	Optionally, specify repetition of the subsequently given vector.
n-times	The number of times to repeat the vector.
dx dy dz	The vector in the global cartesian coordinate system.
dphi	The ‘vector’ in circumferential direction in the specified cylindrical coordinate system, i.e. the spacing in ϕ-direction.
COPY-ONLY	Only copy geometry.
POINT-TO-LINE	Copy geometry and extrude points to lines.
LINE-TO-SURFACE	Copy geometry and extrude points to lines and lines to surfaces.

SURFACE-TO-BODY Copy geometry and extrude points to lines, lines to surfaces and surfaces to bodies.

NOTES:

The geometry names created by the copying and extrusion are as follows:

- Points are named prefixPnm where
  - — prefix is the relevant one of the copy-prefix and extrude-prefix
  - — n is a sequence number referring to the copying/extrusion (1 for the first copy/extrusion 2 for the second and so on)
  - — m is a counter
- Lines are named prefixLnm.
- Surfaces are named prefixSnm.
- Bodies are named prefixBnm.

GENERATE

First specify the level of the geometry to generate and give the id of geometry names:

GENERATE	[BY-NAME]	BODY	id	...
		SURFACE		
		LINE		
		POINT		

Specify the topology space, i.e. the number of points in the three topological directions:

...	I-start	I-end	I-step	I-el	J-start	J-end	J-step	J-el	K-start	K-end	K-step	K-el	...
					END				END				

Choose the type of coordinate system to use in the geometry generation:

...	coord-name									...
	CARTESIAN									
	CYLINDRICAL	orig-x	orig-y	orig-z	zaxi-x	zaxi-y	zaxi-z	raxi-x	raxi-y	raxi-z
	SPHERICAL									

Give the starting point for the generation, i.e. coordinates of the first point:

...	x0	y0	z0	...

Give the vectors for mapping the I-topology into the geometrical coordinate system:

...	[REPEAT	n-times]	{	dxI	dyI	dzI	}*	...
			END					

Give the vectors for mapping the J-topology into the geometrical coordinate system:

...	[REPEAT	n-times]	{	dxJ	dyJ	dzJ	}*	...
			END					

Give the vectors for mapping the K-topology into the geometrical coordinate system:

...	[REPEAT	n-times]	{	dxK	dyK	dzK	}*
			END				

PURPOSE:

The command defines geometry by generating a *regular* geometry consisting of points, lines, surfaces and bodies. A *regular* geometry is (in a cartesian system) a geometry only consisting of surfaces shaped as parallelograms. The command is based on first defining a topological I-J-K space and then, by use of vectors, mapping this space into a geometrical coordinate system.

The GENERATE command is somewhat complex. On the other hand, it is capable of creating large and complex geometries of rectangular, cylindrical and spherical shape with a very limited amount of input. It may therefore be worthwhile to learn how to use it. Section 3.3.7 contains an explanation of the command as well as illustrative examples.

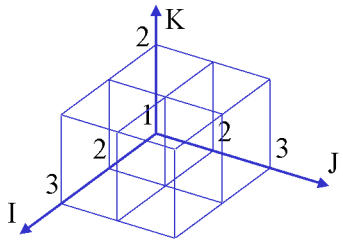


Figure 5.35 Example of a topological space

PARAMETERS:

BY-NAME	This optional command enables adding higher level geometry onto lower level geometry without having to repeat specification of coordinate-system and geometry information. If, for example, points have previously been generated then lines may be added. The GENERATE BY-NAME command will be equal to the original GENERATE command only replacing POINT by LINE and skipping specification of coordinate system, starting-point and vectors.
BODY	Generate bodies, surfaces, lines and points.
SURFACE	Generate surfaces, lines and points.
LINE	Generate lines and points.
POINT	Generate points.
id	Identification of the geometry to generate. Using a single character (A, B, ...) will normally be adequate. See Section 3.3.7 for more details.
I-start I-end I-step	The topology in I-direction. A start value, an end value and a step value defines the range. I-start must be equal to or greater than 1. I-end must be equal to or greater than I-start. I-step must be equal to or greater than 1. The stepping (incrementing) does not need to hit the end value. For example, the start, end and step values: 3, 10 and 2 yield the range 3, 5, 7, 9.
I-el	Number of elements assigned to each line in I-direction.
J-start J-end J-step	The topology in J-direction. See the explanation for the I-direction above.
J-el	Number of elements assigned to each line in J-direction.
K-start K-end K-step	The topology in K-direction. See the explanation for the I-direction above.
K-el	Number of elements assigned to each line in K-direction.

END	By giving END rather than entering topology information for the J-direction the topology space is restricted to 1-D. By giving END rather than entering topology information for the K-direction the topology space is restricted to 2-D.
coord-name	Name of a previously defined coordinate system. This may have been defined by the DEFINE COORDINATE-SYSTEM command or a previous GENERATE command.
CARTESIAN	The cartesian coordinate system of the model is used.
CYLINDRICAL	A cylindrical coordinate system is defined and used. The coordinate system defined is given a name as explained in Section 3.3.7.
SPHERICAL	A spherical coordinate system is defined and used. The coordinate system defined is given a name as explained in Section 3.3.7.
orig-x orig-y orig-z	Cartesian coordinates of the origin of the coordinate system.
zaxi-x zaxi-y zaxi-z	Cartesian coordinates of a point defining the z-axis (for cylindrical coordinate system) or pole axis (for spherical coordinate system).
raxi-x raxi-y raxi-z	Cartesian coordinates of a point defining the ϕ=0-plane (which determines the r-axis).
x0 y0 z0	The cartesian coordinates of the starting point. For cylindrical coordinates r0, ϕ0, z0 are given. For spherical coordinates r0, ϕ0, θ0 are given.
REPEAT	This optional command allows repeating the subsequently given vector a specified number of times.
n-times	The number of times to repeat the subsequently given vector.
dxI dyI dzI	The cartesian components of the I-vectors. For cylindrical coordinates drI, dϕI, dzI are given. For spherical coordinates drI, dϕI, dθI are given. The number of vectors to give corresponds to the topology range. Conclude entering vectors by END. If less vectors than required by the topology range are given then the last vector is repeated the required number of times. If too many vectors are given the superfluous vectors are discarded.
dxJ dyJ dzJ	The cartesian components of the J-vectors. For cylindrical coordinates; drJ, dϕJ, dzJ are given. For spherical coordinates drJ, dϕJ, dθJ are given.
dxK dyK dzK	The cartesian components of the K-vectors. For cylindrical coordinates; drK, dϕK, dzK are given. For spherical coordinates drK, dϕK, dθK are given.

NOTES:

Coordinate systems implicitly defined by the GENERATE command (using the options CYLINDRICAL and SPHERICAL) have the same application area as coordinate systems explicitly defined by the DEFINE COORDINATE-SYSTEM command.

HELP

HELP	SUPPORT
	GENERAL-SYNTAX
	SPECIAL-KEYS
	STATUS-LIST

PURPOSE:

The command provides information on subjects.

PARAMETERS:

SUPPORT — The telephone and telefax numbers and the Internet address for requesting support is printed together with detailed information on the program version used. This information is of interest in connection with support requests. The information is printed in the print window (line-mode window on Unix).

GENERAL-SYNTAX — Information on how to enter commands and text is provided. The information is printed in the print window (line-mode window on Unix).

SPECIAL-KEYS — Information on some special keys is provided. The information is printed in the print window (line-mode window on Unix).

STATUS-LIST — If the program is used in line-mode (Unix only) the Status List is printed on the screen.

If the program is used in graphic input mode the STATUS program is started.

JOIN

JOIN	name1	name2

PURPOSE:

The command joins two neighbouring bodies into a single body. The two bodies must have a common surface which can be either a top, bottom or side surface in either of them. The orientation of the top/bottom surfaces is inherited from the body named first.

PARAMETERS:

name1 Name of a previously defined body to be joined with another body. This will also be the name of the new common body.

name2 Name of a previously defined body to be joined with the first body. This name will disappear.

LABEL

LABEL	ALL-LABELS	...	ON
	BEAM-ELEMENT		
	BODY-NAME		OFF
	BOUNDARY-CONDITION-SYMBOL		
	COLOUR-IDENTIFICATION	...	
	DAMPER-NAMES	...	ON
	ELEMENT-NORMAL		
	ELEMENT-NUMBER		
	ELEMENT-THICKNESS		
	GEOMETRY-NAMES		
	LINE-DIVISIONS		
	LINE-NAME		OFF
	MASS-ELEMENT-NAMES		
	MATERIAL-NAMES		
	MESH-CORNERS		
	NODE-NUMBER		
	NODE-SYMBOL	...	
	POINT-NAME	...	ON
	SECTION-NAMES		
	SPRING-NAMES		
	SUPER-NODE-SYMBOL		OFF
	SURFACE-NAME		
	SURFACE-NORMAL		

PURPOSE:

The command switches on and off certain labels or additional information related to the geometry or element model. A label switched on will appear in the display and remain on for all subsequent displays until switched off. A label switched off will disappear with the next display (or RE-DISPLAY) and remain off until switched on. If a label is currently switched off then ON will be the default choice of the command (select by hitting return) and vice versa.

The options COLOUR-IDENTIFICATION and NODE-SYMBOL are, however, different in that they are not mere on/off switches. See further explanations for these options.

PARAMETERS:

ALL-LABELS	Switch for all labels relevant for the current display.
BEAM-ELEMENT	This option is no longer active.
BODY-NAME	Body name switch.
BOUNDARY-CONDITION-SYMBOL	Boundary condition symbol switch. See Figure 5.36 for an explanation of the various boundary condition symbols.
COLOUR-IDENTIFICATION	Switch for colour identification of either element thickness or material name assigned. See a separate description of this command.
DAMPER-NAMES	Damper name switch.
ELEMENT-NORMAL	Switch for the positive direction of the element local z-axis for 2-D elements as well as the local z-axis of beam elements. Only relevant when the FE mesh is displayed.
ELEMENT-NUMBER	Element number switch.
ELEMENT-THICKNESS	Switch for thickness of 2-D elements. The thickness may be shown numerically or symbolically; see the command SET GRAPHICS PRESENTATION ELEMENT-THICKNESS.
GEOMETRY-NAMES	Joint switch for point, line, surface and body names.
LINE-DIVISIONS	Switch for the number of elements the lines are divided into. See SET NUMBEROF-ELEMENTS and SET MAX-ELEMENT-LENGTH.
LINE-NAME	Line name switch.
MASS-ELEMENT-NAMES	Mass element name switch.
MATERIAL-NAMES	Material name switch.
MESH-CORNERS	Mesh-corner symbol switch.See Figure 5.36.
NODE-NUMBER	Node number switch.
NODE-SYMBOL	Node symbol switch. See Figure 5.36 for an explanation of the various node symbols.
POINT-NAME	Point name switch.
SECTION-NAMES	Section name switch.
SPRING-NAMES	Spring name switch.
SUPER-NODE-SYMBOL	Supernode symbol switch. See Figure 5.36.

SURFACE-NAME	Surface name switch.
SURFACE-NORMAL	Surface normal switch.

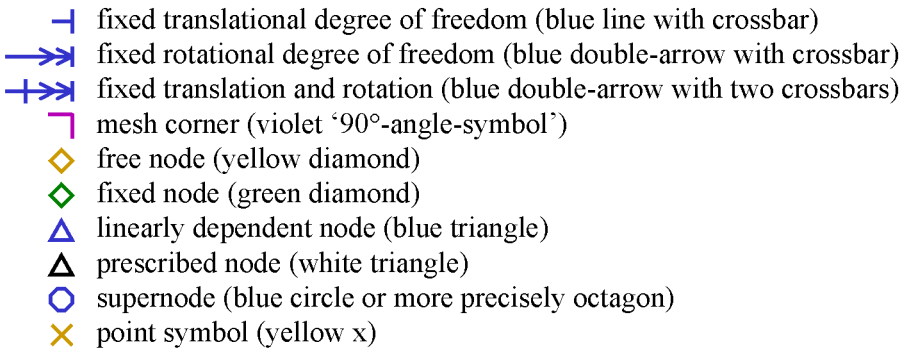

fixed translational degree of freedom (blue line with crossbar)
fixed rotational degree of freedom (blue double-arrow with crossbar)
fixed translation and rotation (blue double-arrow with two crossbars)
mesh corner (violet '90°-angle-symbol')
free node (yellow diamond)
fixed node (green diamond)
linearly dependent node (blue triangle)
prescribed node (white triangle)
supernode (blue circle or more precisely octagon)
point symbol (yellow x)

Figure 5.36 Symbols

NOTES:

The size of the symbols and names is controlled by the command SET GRAPHICS SIZE-SYMBOLS.

Element thicknesses, eccentricities and section outlines are multiplied by a factor in order to be visible on the screen. This factor can be changed using the SET GRAPHICS SIZE-SYMBOLS SECTION-FACTOR command. The factor may be set to values between 1 and 100. Loads drawn with their eccentricities will also use this factor.

LABEL COLOUR-IDENTIFICATION

...	COLOUR-IDENTIFICATION	THICKNESS	DISCRETE-VALUES	ABOVE	value	tol
			SPECIFIED-RANGES	value	*	
				END		
		MATERIAL				
		OFF				

PURPOSE:

The command shows by colours the thickness of 2-D elements or the material name of 2-D and 3-D elements. Up to 15 different colours are available. The thicknesses and material types cannot be shown at the same time. The command is best used in combination with the SET GRAPHICS PRESENTATION FILLED-ELEMENTS ON command. By default, only the edges of the elements are drawn in colour.

A simplified way of drawing in hidden mode is employed implying that the SET GRAPHICS HIDDEN ON command normally need not be used.

PARAMETERS:

THICKNESS — Colour 2-D elements according to their thicknesses. Only one colour is used for each element. When the thickness varies within an element the thickness in one of the nodes is shown for the whole element.

DISCRETE-VALUES — All discrete thickness values are coloured above a given value. A tolerance is used to determine whether two (slightly) different values are to be presented as the same or not. This option is convenient when different constant values have been specified for the element thicknesses.

ABOVE — Only the thicknesses above the subsequently given value are presented.

SPECIFIED-RANGES — The thicknesses in between given values are coloured. Several values (similar to contour lines or iso-curves) may be entered. The END alternative concludes the list of values. This option is convenient when variable thickness has been specified.

value — Thickness values determining the presentation.

tol — Tolerance value determining whether two thicknesses are the same or not.

MATERIAL — Colour 2-D and 3-D elements according to the material name assigned (connected).

OFF — Switch off colour identification of either thickness or material.

NOTES:

In graphic input mode there are Shortcut commands, see Section 3.1, available for presenting the thicknesses and material names using the filled-element option.

LABEL NODE-SYMBOL

		ON		
		OFF		
...	NODE-SYMBOL	FIXED-NODES	...	ON
		FREE-NODES		
		LINEAR-DEPENDENT-NODES		OFF
		PRESCRIBED-NODES		
		SUPER-NODES		

PURPOSE:

The command switches on and off labels (symbols) for nodes according to their boundary condition. The node symbols are shown in Figure 5.36. The symbols for fixed degrees of freedom (line and double-arrow with crossbars) are not switched on by this command but rather the LABEL BOUNDARY-CONDITION-SYMBOL command.

In addition to switching on and off the node symbols individually they can all be switched on and off by LABEL NODE-SYMBOL ON/OFF.

PARAMETERS:

FIXED-NODES	Switch on or off symbol for fixed nodes (green diamond).
FREE-NODES	Switch on or off symbol for free nodes (yellow diamond).
LINEAR-DEPENDENT-NODES	Switch on or off symbol for linearly dependent nodes (blue triangle).
PRESCRIBED-NODES	Switch on or off symbol for prescribed nodes whether it be prescribed displacements or prescribed accelerations (white triangle).
SUPER-NODES	Switch on or off symbol for supernodes (blue circle or more precisely octagon).

NOTES:

If a node has a combination of boundary conditions the symbol chosen for the node will be according to the following precedence: supernode, linearly dependent, prescribed, fixed, free. The number of degrees of freedom having a particular boundary condition is of no importance for the choice of symbol. E.g. a node may have five degrees of freedom fixed but if only a single degree of freedom is super then the node will be labelled as a supernode.

LOCATE

LOCATE

PURPOSE:

The command adds the geometry using dotted lines to the current display. The LOCATE command is useful for locating the currently displayed part of the complete model (either a geometry display or mesh display).

MESH

MESH	name-of-geometry	*
	ADJUST	select-surfaces
	ALL	
	CRACK[a]	
	PART	select-geometry
		ALL-SPRINGS-DAMPERS-INCLUDED

a. This option is presently inactive.

PURPOSE:

The command creates the FE mesh plus damper and spring elements. The command is equivalent to the CREATE MESH command.

The MESH ADJUST command, however, does not create a mesh, rather it adjusts the element division on lines making non-meshable surfaces meshable. (A surface is non-meshable if Prefem is incapable of creating a mesh on it, which again may be because the transition from fine to coarse mesh is too abrupt.) Note that the command merely makes the surfaces meshable, the resulting mesh may be distorted and therefore unacceptable. The command may be used before or after meshing the geometry.

It should be noted that the process of adjusting the element division will normally affect surfaces neighbouring the selected surfaces causing the mesh for these neighbour surfaces to be deleted. In the general case, any surface of the model may be affected and its mesh deleted.

Note: The mesh-adjusting function is implemented for surfaces only.

PARAMETERS:

name-of-geometry — Give the names of lines, surfaces or bodies to be meshed. Wildcard selection may be used, see Section 3.9.3, and pre-defined sets may be used, see Section 3.9.4.

Spring and damper element names (see the DEFINE SPRING/DAMPER commands) may also be given involving such elements to be created.

ADJUST — Adjust element division on lines to make non-meshable surfaces meshable. See explanation above.

select-surfaces — Surfaces that shall be made meshable. See Section 5.1 on how to perform a selection.

ALL — Mesh all geometric entities for which type of element has been defined.

PART	Mesh only selected geometry or create all spring and damper elements.
select-geometry	Geometry to be meshed. See Section 5.1 on how to perform a selection.
ALL-SPRINGS-DAMPERS-INCLUDED	Create all spring and damper elements.

PLOT

PLOT	DISPLAYED			textline	*4	A1
	GEOMETRY					
	MESH					A2
	BODY	select-geometry				A3
	SURFACE					
	LINE					
	POINT					A4
	ELEMENT	select-elements				
	NODE	select-nodes				A5
	LOAD	load-case	load-type			

PURPOSE:

The command generates a plot file which subsequently can be sent to a printer or imported in a word processor. The drawing is plotted with the current viewing position.

Four lines of text, each of maximum 24 characters, are specified. These are reproduced on the plot.

The date and time when the plot was generated will be given on the plot together with the scale.

PARAMETERS:

DISPLAYED	Plot the current display.
GEOMETRY	Plot the whole geometry.
MESH	Plot the whole element mesh.
BODY	Plot selected bodies.
SURFACE	Plot selected surfaces.
LINE	Plot selected lines.
POINT	Plot selected points.
select-geometry	Select geometry to be plotted. See Section 5.1 on how to perform a selection.
ELEMENT	Plot selected elements.
select-elements	Select elements to be plotted. See Section 5.2 on how to perform a selection.
NODE	Plot selected nodes.

select-nodes	Select nodes to be plotted. See Section 5.2 on how to perform a selection.
LOAD	Plot a selected load together with the element mesh. See the ADD-DISPLAY command for an explanation of the graphic presentation of the loads.
load-case	The load case to plot. If a load case contains several load types (line load, normal pressure, etc.) then only one of the load types may be plotted at a time.
load-type	Type of load for the load case to plot. See the PROPERTY LOAD command for the different types of loads.
textline	A line of text of maximum 24 characters. If including blanks the line must be enclosed by single quotes: 'This is an example.'
A1 - A5	European standard paper formats (sizes). A4 is approximately 21 × 30 cm (8.3 × 11.7 inches). A1, A2, etc. have all the same width/height relation being such that when cut in half (parallel with the shortest side) the width/height relation remains the same. A1 is by definition 1 m^2, A2 is half of A1, A3 is half of A2, etc. This entry is dummy for other plot formats than SESAM-NEUTRAL.

NOTES:

The default file name and prefix of the plot file are the same as for the model file. This may be changed by the SET PLOT FILE command (the SET GRAPHIC PLOT-FILE command has the same effect).

The default format is SESAM-NEUTRAL. This may be changed to PostScript, CGM and other formats by the SET PLOT FORMAT command.

The SET PLOT COLOUR command is used to store colour information on the plot file.

When plotting colours you may want to adjust certain light colours to make them visible on white paper. Use the SET GRAPHICS COLOUR command for this purpose.

If plot properties are to be set by the SET PLOT command these settings must be done prior to giving the PLOT command.

The paper format (A1 - A5) is only relevant for the SESAM-NEUTRAL plot format. For other formats this entry is dummy.

If 2 or 4 viewports are used then only the isometric view is plotted.

PRINT

PRINT	ALL					
	STATUS					
	BODY	select-geometry	GEOMETRY			
	SURFACE					
	LINE		PROPERTY			
	POINT					
	SET	set-name				
		ALL				
	SHAPE	select-shapes				
	SPRING-DAMPER	select-spring-dampers				
	ELEMENT	select-elements	BASIC-ELEMENT			
			ECCENTRICITY			
			LOCAL-COORDINATE-SYSTEM			
	NODE	select-nodes	COORDINATES			
			BOUNDARY-CONDITIONS			
			LINEAR-DEPENDENCY			
			INITIAL-CONDITIONS			
			MASS			
	LOAD	load-case	...	ALL-LOADTYPES	select-geometry	
		ALL		load-type		
	LAYERED		...	name		
	MATERIAL			ALL		
	SECTION					
	TRANSFORMATION		trnam	...	DECODED	
			ALL		UNDECODED	
	CONNECTOR[a]					

a. This option is presently inactive.

PURPOSE:

The command prints selected information on the screen or to a print file.

PARAMETERS:

ALL	Generate a print file containing a complete printout of all defined and created data.
STATUS	Print key information on the model.
BODY	Print information on selected bodies.
SURFACE	Print information on selected surfaces.
LINE	Print information on selected lines.
POINT	Print information on selected points.
select-geometry	Select geometry to be printed. See Section 5.1 on how to perform a selection.
GEOMETRY	Print geometrical information.
PROPERTY	Print property type information.
SET	Print information on selected sets.
set-name	Name of a set name.
SHAPE	Print information on selected shapes.
select-shapes	Shapes are selected in a similar way as geometry; see Section 5.1. Wild-card may, however, not be used for selecting shapes; their names must be given explicitly.
SPRING-DAMPER	Print information on selected spring and/or damper elements.
select-spring-dampers	The spring and damper elements are selected in a similar way as geometry; see Section 5.1. Wild-card may, however, not be used for selecting shapes; their names must be given explicitly.
ELEMENT	Print information on selected elements.
select-elements	Select elements for printing. See Section 5.2 on how to perform a selection.
BASIC-ELEMENT	Print basic element type information.
ECCENTRICITY	Print information on element eccentricities.
LOCAL-COORDINATE-SYSTEM	Print information on the element local coordinate systems.
NODE	Print information on selected nodes.
select-nodes	Select nodes for printing. See Section 5.2 on how to perform a selection.

COORDINATES	Print nodal coordinates.
BOUNDARY-CONDITIONS	Print nodal boundary conditions.
LINEAR-DEPENDENCY	Print linear dependencies of nodes.
INITIAL-CONDITIONS	Print initial conditions of nodes.
MASS	Print nodal masses.
LOAD	Print information on selected loads.
load-case	Select load case for printing.
ALL	Select all load cases for printing.
ALL-LOAD-TYPES	Print information on all load types for selected load cases.
load-type	Type of load for the selected load cases for which to print information. See the PROPERTY LOAD command for the different types of loads.
LAYERED	Print information on selected layered names.
MATERIAL	Print information on selected material names.
SECTION	Print information on selected section names.
name	Name of layered/material/section to be printed.
TRANSFORMATION	Print information on selected transformation names.
trnam	Transformation name to print.
DECODED	Print the transformations in decoded form, i.e. in terms of translations and rotations.
UNDECODED	Print the transformations in un-decoded form, i.e. print the transformation matrices.

NOTES:

Information printed on the screen is broken into a certain number of lines in order not to fill more than a single window at a time. The user is requested to CONTINUE or END the printing after each screen-fill. As the CONTINUE alternative is the default choice a semicolon (;) accepting all subsequent default values involves printing all information.

The PRINT ALL command will always generate a print file. All other PRINT commands will by default direct the print to the screen. The SET PRINT DESTINATION command may change this so that all print goes to a print file.

Some typical print tables are presented below with explanations of their contents.

Print of point coordinates

The point coordinates are tabulated.

POINT NAME	X	Y	Z
P1	0.00000E+00	0.00000E+00	0.00000E+00
P2	1.00000E+01	0.00000E+00	0.00000E+00
P3	1.20000E+01	6.00000E+00	0.00000E+00
P4	8.00000E+00	7.00000E+00	0.00000E+00
P5	4.00000E+00	7.00000E+00	0.00000E+00
P6	0.00000E+00	7.00000E+00	0.00000E+00
PCENTRE	9.00000E+00	3.50000E+00	0.00000E+00

Print of point properties

The table contains point masses. There are two lines for each point: the first line is translational masses while the second is rotational masses. The column 'BOUND' contains a six character boundary condition code, one character for each of the six degrees of freedom: F=fixed, P=prescribed, L=linearly dependent, S=super. The column 'LIN-DEPENDENCY BY POINT' contains information on linear dependencies.

POINT NAME	MASS X	MASS Y	MASS Z	BOUND	LIN-DEPENDENCY BY POINT
P1				FFF F	
P2				FFF F	
P3				PPP	
P4	350.000000	350.000000	350.000000		----
	0.000000	0.000000	0.000000		
P5	350.000000	350.000000	350.000000	LLLLLL	T-P P6
	0.000000	0.000000	0.000000		P4
P6				SSS	----

Print of geometry information for lines

The table tells by which points the lines, arcs and curves are defined. If a line/curve is defined as the intersection between shapes then the shape names are given as well. The number of elements that the lines, arcs and curves will be divided into is also given (column 'PARTS').

LINE NAME	TYPE	PARTS	START POINT	END POINT	CENTER /POINT	NEAR.PO. /POINT	SHAPE1 /POINT	SHAPE2 /POINT
L1	LINE	8	P1	P2				
L2	LINE	4	P2	P3				
L3	LINE	3	P3	P4				
L4	LINE	3	P4	P5				
L5	LINE	2	P5	P6				
ARC1	ARC	4	P6	P1	PCENTRE			

Print of property information for lines

The table contains property information for lines: type of element, material, boundary condition code (see 'Print of point properties above'), type of cross section, existence of load (an 'x' in column 'LOAD' informs that a load has been defined for the line) and eccentricity ('CALC' means that the eccentricity is calculated by the program, i.e. the PROPERTY ECCENTRICITY-BEAM ... CALCULATED-... command has been used).

LINE NAME	ELEMENT TYPE	MATERIAL NAME	BOUND. COND.	TRANSF. NAME	CROSS SECTION	LOAD	ECCEN-TRICITY	
L1		STEEL	FFF F					
L2	BEAM-2	STEEL			BOX	x	LOCAL:	
							X=	0.400
							Z=	-1.000
L3		STEEL						
L4	BEAM-2	STEEL			LSECT		CALC	
L5	BEAM-2	STEEL			LSECT		CALC	
ARC1	BEAM-2	STEEL			LSECT		CALC	

Print of geometry information for surfaces

The table tells by which lines, arcs and curves the surfaces have been defined. A plus sign (+) preceding a line name informs that there is a so-called 'not-mesh-corner' between that line and the preceding line in the list.

SURFACE NAME	SHAPE NAME	LINES					
S1		L1	L2	L3	+L4	+L5	ARC1

Print of property information for surfaces

The table contains property information for surfaces: type of element, material, boundary condition code (see 'Print of point properties above'), existence of load (an 'x' in column 'LOAD' informs that a load has been defined for the line) and thickness.

SURFACE NAME	ELEMENT TYPE	MATERIAL NAME	BOUND. COND.	TRANSF. NAME	LOAD	THICKNESS
S1	SHELL-4	STEEL			x	5.00000E-02

Print of material data

All material types are printed.

```
MATERIAL NAME   :  STEEL
MATERIAL NUMBER :            1
MATERIAL TYPE   :  Linear isotropic elastic, structural analysis
Young's modulus                                          2.1000E+11
Poisson's ratio                                          3.0000E-01
Density                                                  7.8500E+03
Thermal expansion coefficient                            1.2000E-05
```

Print of cross sectional data

All cross section types are printed. In addition to the given data (cross sectional geometry data) computed data are printed.

```
SECTION NAME   :   LSECT
SECTION NUMBER :            1
SECTION TYPE   :   L-SECTION

HZI     HEIGHT AT END                                                0.800000
TY      WEB THICKNESS                                                0.030000
BY      FLANGE WIDTH                                                 0.400000
TZ      FLANGE THICKNESS                                             0.030000
SFY     SHEAR FACTOR Y DIRECTION                                     1.000000
SFZ     SHEAR FACTOR Z DIRECTION                                     1.000000
K       WEB LOCATION IN LOCAL Y-DIRECTION                 POSITIVE

AREA    CROSS SECTION AREA                                           0.035100
IX      TORSIONAL MOMENT OF INERTIA ABOUT SHEAR CENTRE               0.000010
IY      MOMENT OF INERTIA ABOUT Y AXIS                               0.002406
IZ      MOMENT OF INERTIA ABOUT Z AXIS                               0.000432
IYZ     PRODUCT OF INERTIA ABOUT Y AND Z AXES                        0.000584
WXMIN   MIN. TORSIONAL SECTION MODULUS ABOUT SHEAR CENTRE            0.000295
WYMIN   MIN. SECTION MODULUS ABOUT Y AXIS                            0.004611
WZMIN   MIN. SECTION MODULUS ABOUT Z AXIS                            0.001343
SHARY   SHEAR AREA IN THE DIRECTION OF Y AXIS                        0.008346
SHARZ   SHEAR AREA IN THE DIRECTION OF Z AXIS                        0.017675
SHCENY  SHEAR CENTRE LOCATION FROM CENTROID Y COMPONENT              0.063248
SHCENZ  SHEAR CENTRE LOCATION FROM CENTROID Z COMPONENT             -0.263248
SY      STATIC AREA MOMENT ABOUT Y AXIS                              0.004083
SZ      STATIC AREA MOMENT ABOUT Z AXIS                              0.001553
CY      CENTROID LOC. FROM BOTTOM RIGHT CORNER Y COMPONENT           0.321752
CZ      CENTROID LOC. FROM BOTTOM RIGHT CORNER Z COMPONENT           0.278248


SECTION NAME   :   BOX
SECTION NUMBER :            2
SECTION TYPE   :   BOX

HZI     HEIGHT AT END                                                0.600000
BY      SECTION WIDTH                                                0.400000
TT      UPPER WALL THICKNESS                                         0.020000
TY      SIDE WALL THICKNESS                                          0.020000
TB      LOWER WALL THICKNESS                                         0.020000
SFY     SHEAR FACTOR Y DIRECTION                                     1.000000
SFZ     SHEAR FACTOR Z DIRECTION                                     1.000000

AREA    CROSS SECTION AREA                                           0.038400
IX      TORSIONAL MOMENT OF INERTIA ABOUT SHEAR CENTRE               0.002024
IY      MOMENT OF INERTIA ABOUT Y AXIS                               0.001932
IZ      MOMENT OF INERTIA ABOUT Z AXIS                               0.001023
IYZ     PRODUCT OF INERTIA ABOUT Y AND Z AXES                        0.000000
```

```
WXMIN   MIN. TORSIONAL SECTION MODULUS ABOUT SHEAR CENTRE            0.008816
WYMIN   MIN. SECTION MODULUS ABOUT Y AXIS                            0.006438
WZMIN   MIN. SECTION MODULUS ABOUT Z AXIS                            0.005114
SHARY   SHEAR AREA IN THE DIRECTION OF Y AXIS                        0.013972
SHARZ   SHEAR AREA IN THE DIRECTION OF Z AXIS                        0.019872
SHCENY  SHEAR CENTRE LOCATION FROM CENTROID Y COMPONENT              0.000000
SHCENZ  SHEAR CENTRE LOCATION FROM CENTROID Z COMPONENT              0.000000
SY      STATIC AREA MOMENT ABOUT Y AXIS                              0.003888
SZ      STATIC AREA MOMENT ABOUT Z AXIS                              0.002928
CY      CENTROID LOC. FROM BOTTOM RIGHT CORNER Y COMPONENT           0.200000
CZ      CENTROID LOC. FROM BOTTOM RIGHT CORNER Z COMPONENT           0.300000
```

Print of loads

The various types of loads are tabulated for the geometry for which they have been defined.

```
                                  CONCENTRATED LOADS
                                  ==================

LOAD CASE NUMBER  :         1
GEOMERTY      COORD    TRANS
           Fx/Fr       Fy/Fphi     Fz/Ftheta      Mx/Mr       My/Mphi     Mz/Mtheta
         ----------- ----------- ----------- ----------- ----------- -----------
P3
         0.00000E+00 0.00000E+00 -8.0000E+02 0.00000E+00 0.00000E+00 0.00000E+00

                                  DISTRIBUTED LINE LOAD
                                  =====================

 LOAD CASE NUMBER  :         2
 LINE    COORD    TRANSF
 NAME    SYSTEM   NAME             Fx/Fr            Fy/Fphi        Fz/Ftheta   POS
-------- -------- -------- --- -------------- -------------- -------------- ---
L2                        REA    9.00000E+01   -5.00000E+01   -4.00000E+01 MID
                                  NORMAL PRESSURE LOADS
                                  ====================

LOAD CASE NUMBER  :         3
SURFACE
 NAME       PRESSURE      IMAG. PART.      SIDE
-------- ------------- ------------- ---------
S1           8.5000E+00                MIDSIDE
```

Print of nodal coordinates and other information

Nodal coordinates are tabulated. The ‘X’ in column ‘BOU CON’ informs that a boundary condition has been defined for that node (see the next table for details). The number in the column ‘ND’ is the number of degrees of freedom for the node.

```
NODE                 C O O R D I N A T E S             BOU
NO.             X              Y              Z        CON ND
------- -------------- -------------- -------------- --- --
      1      10.000000       0.000000       0.000000  X   6
      2      10.500000       1.500000       0.000000      6
```

```
       3      11.000000       3.000000       0.000000      6
       4      11.500000       4.500000       0.000000      6
       5      12.000000       6.000000       0.000000  X   6
       6       8.000000       7.000000       0.000000      6
etc.
```

Print of boundary conditions for nodes

The table gives details on nodal boundary conditions.

```
 NODE     TRANSF             BOUNDARY CONDITIONS
 NO.      NO.       TX     TY     TZ     RX     RY     RZ
 -------  ------- ------ ------ ------ ------ ------ ------
       1          FIXED  FIXED  FIXED                FIXED
       5          FIXED  FIXED  FIXED                FIXED
       9          PRESC. PRESC. PRESC.
      11          SUPER  SUPER  SUPER
      15          LINEAR LINEAR LINEAR LINEAR LINEAR LINEAR
      16          FIXED  FIXED  FIXED                FIXED
etc.
```

Print of element information

The table provides details on the elements. The column ‘THICKNESS/SECT. NO.’ informs about the thickness of membrane and shell elements and cross section number for beam elements (the ‘Print of cross sectional data’ shows that a number is assigned to each cross section name).

```
 ELEMENT  ELEMENT  MATERIAL  THICKNESS
   NO.     TYPE       NO.    /SECT. NO.                NODE NUMBER
 -------  -------  --------  ---------   ------- ------- ------- ------- -------
      1   BEAS                      2         1       2
      2   BEAS                      2         2       3
      3   BEAS                      2         3       4
      4   BEAS                      2         4       5
      5   BEAS                      1         6       7
      6   BEAS                      1         7       8
      7   BEAS                      1         8       9
      8   BEAS                      1         9      10
      9   BEAS                      1        10      11
     10   BEAS                      1        11      12
     11   BEAS                      1        12      13
     12   BEAS                      1        13      14
     13   BEAS                      1        14      15
     14   FQUS              5.00E-02        15      16      25      14
     15   FQUS              5.00E-02        16      17      26      25
     16   FQUS              5.00E-02        17      18      27      26
     17   FQUS              5.00E-02        18      19      28      27
etc.
```

Print of element local coordinate systems

The table informs about the local coordinate systems of all elements. The cosines relative to the coordinate system of the superelement is given for each of the local axes.

EXT.		LOCAL-X			LOCAL-Y			LOCAL-Z			
EL.	ND	GX	GY	GZ	GX	GY	GZ	GX	GY	GZ	REM.
1	1	0.316	0.949		-0.949	0.316				1.000	SPEC.
2	1	0.316	0.949		-0.949	0.316				1.000	SPEC.
3	1	0.316	0.949		-0.949	0.316				1.000	SPEC.
4	1	0.316	0.949		-0.949	0.316				1.000	SPEC.
5	1	-1.000				-1.000				1.000	SPEC.
6	1	-1.000				-1.000				1.000	SPEC.
7	1	-1.000				-1.000				1.000	SPEC.
8	1	-1.000				-1.000				1.000	SPEC.
9	1	-1.000				-1.000				1.000	SPEC.
10	1	-0.275	-0.962		0.962	-0.275				1.000	SPEC.
11	1	-0.093	-0.996		0.996	-0.093				1.000	SPEC.
12	1	0.093	-0.996		0.996	0.093				1.000	SPEC.
13	1	0.275	-0.962		0.962	0.275				1.000	SPEC.
14											NONE
15											NONE
16											NONE

etc.

Print of element eccentricities

The table provides details on eccentricities (offsets) for the elements.

OFFSETS IN SUPERELEMENT'S COORDINATE SYSTEM, FROM NODE TO ELEMENT

EXT.	INT.	ECCENTRICITIES AT ODD NODES			ECCENTRICITIES AT EVEN NODES		
EL.	EL.	X	Y	Z	X	Y	Z
1	1	0.126	0.379	-1.000	0.126	0.379	-1.000
2	2	0.126	0.379	-1.000	0.126	0.379	-1.000
3	3	0.126	0.379	-1.000	0.126	0.379	-1.000
4	4	0.126	0.379	-1.000	0.126	0.379	-1.000
5	5	0.000	0.063	-0.547	0.000	0.063	-0.547
6	6	0.000	0.063	-0.547	0.000	0.063	-0.547
7	7	0.000	0.063	-0.547	0.000	0.063	-0.547
8	8	0.000	0.063	-0.547	0.000	0.063	-0.547
9	9	0.000	0.063	-0.547	0.000	0.063	-0.547
10	10	-0.061	0.017	-0.547	-0.061	0.017	-0.547
11	11	-0.063	0.006	-0.547	-0.063	0.006	-0.547
12	12	-0.063	-0.006	-0.547	-0.063	-0.006	-0.547
13	13	-0.061	-0.017	-0.547	-0.061	-0.017	-0.547

PROPERTY

PROPERTY	BOUNDARY-CONDITION	...
	CHANGE	
	ECCENTRICITY-BEAM	
	HINGE	
	INITIAL-DISPLACEMENT	
	INITIAL-VELOCITY	
	LINEAR-DEPENDENCY	
	LOAD	
	LOCAL-COORDINATE-BEAM	
	LOCAL-COORDINATE-SURFACE	
	MATERIAL	
	POINT-MASS	
	SECTION	
	THICKNESS	
	TRANSFORMATION	

PURPOSE:

The command defines properties such as thicknesses, materials, boundary conditions, loads, etc.

The properties are assigned to geometric entities such as points, lines, surfaces and bodies. The properties are automatically transferred to the element mesh, i.e. the nodes and elements.

PARAMETERS:

BOUNDARY-CONDITION	Define boundary conditions (fixations of nodes).
CHANGE	Change properties previously defined.
ECCENTRICITY-BEAM	Define eccentricities (offsets) for beams.
HINGE	Define hinges for beams.
INITIAL-DISPLACEMENT	Define initial displacements for a dynamic analysis.
INITIAL-VELOCITY	Define initial velocities for a dynamic analysis.
LINEAR-DEPENDENCY	Define linear dependencies between nodes (multi-point constraints).

LOAD	Define loading conditions.
LOCAL-COORDINATE-BEAM	Define local coordinate systems for beam elements.
LOCAL-COORDINATE-SURFACE	Define local coordinate systems for layered elements.
MATERIAL	Define material properties.
POINT-MASS	Define point masses in nodes.
SECTION	Define cross sections.
THICKNESS	Define shell and membrane element thicknesses.
TRANSFORMATION	Define transformations.

PROPERTY BOUNDARY-CONDITION

...	BOUNDARY-CONDITION	select-geometry	...
		select-nodes	

...	FIX	*6	...
	FREE		
	PRESCRIBED-ACCELERATION		
	PRESCRIBED-DISPLACEMENT		
	SUPERNODE		

...	GLOBAL		
	TRANSFORMED	trnam	
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED
			trnam

PURPOSE:

The command defines boundary conditions of the nodes in a selected part of the geometry or for selected nodes directly. Boundary conditions are always specified for 6 degrees of freedom (d.o.f.s). If a node has less than 6 d.o.f.s (solid and membrane elements have only 3 d.o.f.s) the superfluous boundary condition specifications are ignored. By default, the boundary condition of all nodes is FREE.

The following boundary conditions can be given (in parentheses: the code used in print tables):

FREE	(blank)	Free to move
FIXED	(FIXED)	Fixed at zero displacement
PRESCRIBED-DISPLACEMENT	(PRESC.)	Will be given a prescribed displacement
PRESCRIBED-ACCELERATION	(PRESC.)	Will be given a prescribed acceleration
SUPERNODE	(SUPER)	Super d.o.f.

In addition, linear dependencies will introduce boundary conditions; see the command for this.

For the PRESCRIBED-DISPLACEMENT/ACCELERATION the actual displacement/acceleration is given as a load using the command PROPERTY LOAD load-case PRESCRIBED-DISPLACEMENT/ACCELERATION. If no displacement/acceleration is specified then the boundary condition PRESCRIBED-DISPLACEMENT/ACCELERATION is equivalent to a fixation.

If the model is a superelement to be coupled with other superelements then SUPERNODE must be specified for the d.o.f.s to couple.

If the superelement is to be rotated or mirrored then either all 3 translational d.o.f.s, all 3 rotational d.o.f.s or all 6 d.o.f.s must be specified as super. The reason for this is that rotation and mirroring of a superelement

involve multiplying its stiffness matrix by 3 by 3 transformation matrices. Such multiplication cannot be done unless the said requirement is fulfilled.

PARAMETERS:

select-geometry	Select geometry. See Section 5.1 on how to perform a selection.
select-nodes	Select nodes directly. See Section 5.2 on how to perform a selection.
GLOBAL	The boundary conditions refer to the cartesian coordinate system of the model.
TRANSFORMED	The boundary conditions refer to a previously defined transformation of the cartesian coordinate system; see the command DEFINE TRANSFORMATION.
LOCAL-COORDINATE-SYSTEM	The boundary conditions refer to a previously defined (cylindrical or spherical) coordinate system, see the command DEFINE COORDINATE-SYSTEM.
trnam	Name of a previously defined transformation.
coord-name	Name of a previously defined coordinate system.
UNTRANSFORMED	The (cylindrical or spherical) coordinate system is employed without any further transformation.

NOTES:

Boundary conditions defined for geometry will automatically be transferred to the nodes of the geometry.

Whether boundary conditions are defined before or after creating the mesh has no consequence.

The ultimately given boundary condition will override any previously given boundary condition. For example, if two lines meet in a point and the X-translation only is fixed for the first line and the Y-translation only is fixed for the second line then the node created in the common point will have the Y-translation fixed only (the fixations of the two lines are not ‘added’).

Note that deleting the mesh and re-meshing will discard boundary conditions specified directly for nodes. It is therefore in general best to define boundary conditions (as all other properties) for geometry rather than nodes.

PROPERTY CHANGE

...	CHANGE	BOUNDARY-CONDITION
		INITIAL-DISPLACEMENT
		INITIAL-VELOCITY
		LINEAR-DEPENDENCY
		LOAD
		LOCAL-COORDINATE-BEAM
		MATERIAL
		POINT-MASS
		SECTION
		THICKNESS
		TRANSFORMATION

PURPOSE:

The command changes previously defined properties. The command is equivalent to the CHANGE PROPERTY command. The commands have the same syntax and corresponding interpretation as the PROPERTY command defining the data. Therefore, rather than describing these commands in detail here reference is made to the corresponding PROPERTY command.

NOTES:

The PROPERTY CHANGE LOAD load-case TO-MASS command demands special explanation; see the CHANGE PROPERTY LOAD load-case TO-MASS command.

If changes are made to loads by the PROPERTY CHANGE LOAD command then use the RE-COMPUTE LOADS command to redistribute the loads, otherwise an ADD-DISPLAY LOAD command will not give the correct result. This re-computation will automatically be performed when producing the Input Interface File.

Rather than using the PROPERTY CHANGE TRANSFORMATION command the CHANGE TRANSFORMATION command is recommended as the latter is the more flexible and powerful one.

PROPERTY ECCENTRICITY-BEAM

...	ECCENTRICITY-BEAM	select-lines	CALCULATED-NEGATIVE-Z-OFFSET				
			CALCULATED-POSITIVE-Z-OFFSET				
			LOCAL-COORDINATE-OFFSET		x	y	z
			ONE-END-LOCAL-OFFSET	point			

PURPOSE:

The command defines eccentricities for beam elements.

The eccentricity can be specified in beam element local coordinates or applied automatically by Prefem.

PARAMETERS:

select-lines — Select lines for which beam elements shall have eccentricities. See Section 5.1 on how to perform a selection.

CALCULATED-NEGATIVE-Z-OFFSET — The eccentricity is automatically determined to let the beam section be welded or attached onto the surface of the plate or shell. Both the beam sectional data and plate/shell thickness are taken into account; these data must therefore have been defined already. The beam is given eccentricity (shifted) in the negative beam local z-direction; see Figure 5.37. The eccentricity is constant along each line unless the plate/shell thickness varies.

CALCULATED-POSITIVE-Z-OFFSET — Same as CALCULATED-NEGATIVE-Z-OFFSET only the beam is given eccentricity (shifted) in the positive beam local z-direction.

LOCAL-COORDINATE-OFFSET — The eccentricity is specified in beam element local coordinates as a vector from the nodes to the beam ends (and midpoints for 3 node beam elements). The eccentricity is constant along the beam.

ONE-END-LOCAL-OFFSET — The eccentricity is specified in beam element local coordinates for one end of the beam. The eccentricity is a vector from the nodes to the beam ends The beam end is selected by selecting appropriate point.

If several lines have been selected wild-card may be used to select the corresponding points. The line names together with the wild-card point name identify the appropriate points. If the wild-card point name does not match an end point of a selected line the eccentricity is not applied. An error message will be given if the wild-card point name does not match any line ends.

point — (Wild-card) point name.

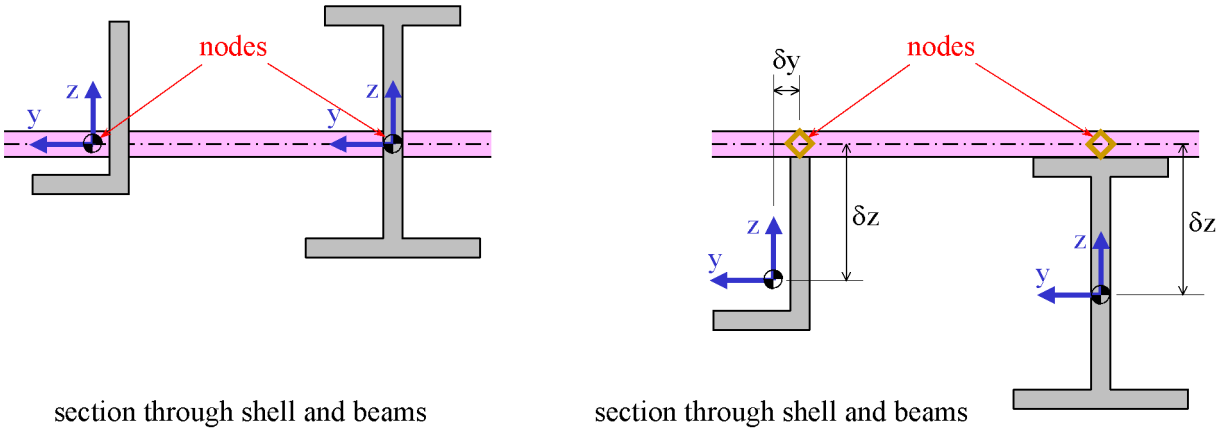
x y z	Eccentricity in beam element local directions pointing from the node to the beam end. Functions as described in Section 5.3 (except for VALUE-BETWEEN and ONLY-BETWEEN) may be used.

NOTES:

If more than one PROPERTY ECCENTRICITY-BEAM command is given for the same line then the last one of the commands CALCULATED-NEGATIVE-Z-OFFSET, CALCULATED-POSITIVE-Z-OFFSET and LOCAL-COORDINATE-OFFSET will take effect. Eccentricities given by the ONE-END-OFFSET command, however, will be added (but only the last ONE-END-LOCAL-OFFSET if there is more than one for the point).

The combined result of PROPERTY ECCENTRICITY-BEAM commands is printed by the PRINT ELEMENT select-element ECCENTRICITY command.

Use of the CALCULATED-NEGATIVE-Z-OFFSET and CALCULATED-POSITIVE-Z-OFFSET options assumes that the plate thickness and cross section have already been defined (the latter must also be assigned or connected to the relevant lines). Having followed this procedure and automatically calculated the eccentricities neither changing the plate thickness nor deleting the plate elements will have any direct effect on the eccentricities. The beam elements must be deleted and re-created or the PROPERTY ECCENTRICITY-BEAM command must be issued once more to change the eccentricities.



section through shell and beams with no eccentricities

section through shell and beams after CALCULATED-NEGATIVE-Z-OFFSET

Figure 5.37 Automatically calculated eccentricity for L- and I-sections

PROPERTY HINGE

...	HINGE	select-lines	point	...

...	GLOBAL			...
	TRANSFORMED	trnam		
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED	
			trnam	
	LOCAL-BEAM-COORDINATE-SYSTEM			

...	FIXATION	alphai	*6
	STIFFNESS	ci	*6
		INFINITY	

PURPOSE:

The command introduces hinges for two node beam elements. The hinge is defined for one end of a beam element at the time. Any d.o.f. may be defined as a 'hinge' including the translational d.o.f.s. Furthermore, stiffnesses may be attached to the hinges. These stiffnesses are either given as coefficients of fixation (α_i) or as elastic spring stiffnesses (c_i).

The relationship between α_i and c_i is:

$\alpha_i = c_i / (k_{ii} + c_i)$

where k_{ii} is the diagonal term of the stiffness matrix corresponding to d.o.f. number 'i' of the relevant node. This implies that:

$\alpha_i = 0$ involves full release of the d.o.f. from the node.

$\alpha_i = 1$ involves full fixation of the d.o.f. to the node.

The result of the PROPERTY HINGE command may be printed by the PRINT ELEMENT select-element HINGE command.

PARAMETERS:

select-lines	Select lines. See Section 5.1 on how to perform a selection.
point	One of the two end points of the selected line(s). Wild card selection is allowed and must be used if more than one line has been selected. If the wild card specification matches one of the end points of the selected line(s) then the hinge will be inserted. If the wild card selection matches non of or both end points the hinge will not be inserted.
GLOBAL	The hinge conditions refer to the cartesian coordinate system of the model.

TRANSFORMED	The hinge conditions refer to a previously defined transformation of the cartesian coordinate system; see the command DEFINE TRANSFORMATION.
LOCAL-COORDINATE-SYSTEM	The hinge conditions refer to a previously defined (cylindrical or spherical) coordinate system, see the command DEFINE COORDINATE-SYSTEM.
LOCAL-BEAM-COORDINATE-SYSTEM	The hinge conditions refer to the local coordinate system of the element itself.
trnam	Name of a previously defined transformation.
coord-name	Name of a previously defined coordinate system.
UNTRANSFORMED	The (cylindrical or spherical) coordinate system is employed without any further transformation.
FIXATION	The hinge stiffnesses are given as coefficients of fixation.
alphai	The coefficient of fixation of d.o.f. number ‘i’.
STIFFNESS	The hinge stiffnesses are given as elastic spring stiffnesses.
ci	The elastic spring stiffness of d.o.f. number ‘i’.
INFINITY	The elastic spring stiffness of d.o.f. number ‘i’ is infinitely high.

PROPERTY INITIAL-DISPLACEMENT

...	INITIAL-DISPLACEMENT	select-geometry	fx	fy	fz	mx	my	mz	...

...	GLOBAL	
	TRANSFORMED	trnam

PURPOSE:

The command specifies the values of an initial displacement for nodes. This is relevant for a dynamic analysis only.

PARAMETERS:

select-geometry	Select geometry. See Section 5.1 on how to perform a selection.
fx	Initial displacement in the X-direction.
fy	Initial displacement in the Y-direction.
fz	Initial displacement in the Z-direction.
mx	Initial rotation about the X-axis given in radians.
my	Initial rotation about the Y-axis given in radians.
mz	Initial rotation about the Z-axis given in radians.
GLOBAL	The values refer to the cartesian coordinate system of the model.
TRANSFORMED	The values refer to a transformation of the cartesian coordinate system.
trnam	Name of a previously defined transformation.

PROPERTY INITIAL-VELOCITY

...	INITIAL-VELOCITY	select-geometry	fx	fy	fz	mx	my	mz	...

...	GLOBAL	
	TRANSFORMED	trnam

PURPOSE:

The command specifies the values of an initial velocity for nodes. This is relevant for a dynamic analysis only.

PARAMETERS:

select-geometry	Select geometry. See Section 5.1 on how to perform a selection.
fx	Initial velocity in the X-direction.
fy	Initial velocity in the Y-direction.
fz	Initial velocity in the Z-direction.
mx	Initial rotational velocity about the X-axis given in radians per second.
my	Initial rotational velocity about the Y-axis given in radians per second.
mz	Initial rotational velocity about the Z-axis given in radians per second.
GLOBAL	The values refer to the cartesian coordinate system of the model.
TRANSFORMED	The values refer to a transformation of the cartesian coordinate system.
trnam	Name of a previously defined transformation.

PROPERTY LINEAR-DEPENDENCY

...	LINEAR-DEPENDENCY	GENERAL-NODE-DEPENDENCY
		LINE-LINE-DEPENDENCY
		RIGID-BODY-DEPENDENCY
		TWO-NODE-DEPENDENCY
		TWO-POINT-DEPENDENCY

PURPOSE:

The command defines the displacements of selected nodes to be linearly dependent of displacements of other selected nodes. See Section 3.5.8 for some general information on how to use this feature.

PARAMETERS:

GENERAL-NODE-DEPENDENCY — Defines linear dependencies between any degree of freedom (d.o.f.) of a node and any other d.o.f.s of any other nodes. This specification will be lost with a DELETE MESH command.

LINE-LINE-DEPENDENCY — Defines linear dependencies between all or selected d.o.f.s of nodes created on a pair of lines. The pair of lines must have the same location (overlap).

RIGID-BODY-DEPENDENCY — Defines linear dependencies between all or selected d.o.f.s of several nodes and a single node.

TWO-NODE-DEPENDENCY — Defines linear dependencies between one node and two other nodes. The nodes are selected by referring directly to node numbers. This specification will be lost with a DELETE MESH command.

TWO-POINT-DEPENDENCY — Defines linear dependencies between one node and two other nodes. The nodes are selected by referring to geometry points.

PROPERTY LINEAR-DEPENDENCY GENERAL-NODE-DEPENDEN-CY

...	GENERAL-NODE-DEPENDENCY	...

...	dep-node	{	dep-dof	{	indep-node	{	indep-dof	beta	}*	}*	}*
						END					
				END							
		END									

PURPOSE:

The command defines linear dependencies between any degree of freedom (d.o.f.) of a node and any other d.o.f.s of any other nodes. This specification will be lost with a DELETE MESH command.

The braces and asterisks indicate that each d.o.f. of a dependent node may be made linearly dependent of several d.o.f.s of several other nodes.

PARAMETERS:

dep-node — Node number of the dependent node.

dep-dof — Degree of freedom to be dependent. Enter either of the codes X, Y, Z, R-X, R-Y and R-Z (X = translation in X, R-X = rotation about X, etc.).

indep-node — Node number of the independent node.

indep-dof — The independent degree of freedom. Enter either of the codes X, Y, Z, R-X, R-Y and R-Z. In addition to these codes there are codes used for forcing the selected d.o.f. to become so-called ‘superl’. Confer with Section 3.5.8 to select the appropriate code.

beta — Linear dependency factor, i.e. the quotient between the displacement of the dependent d.o.f. and the displacement of the independent d.o.f.

PROPERTY LINEAR-DEPENDENCY LINE-LINE-DEPENDENCY

...	LINE-LINE-DEPENDENCY	dep-line	indep-line	ALL		
				TRANSLATIONS-ONLY		
				SELECTED	DEPENDENT	*6
					NON-DEPENDENT	

PURPOSE:

The command defines linear dependencies between all or selected d.o.f.s of nodes created on a pair of lines. The pair of lines must have the same location (overlap). The command is intended to be used as illustrated in Figure 5.38.

A pair of dependent and independent nodes having the same coordinates will be directly linearly coupled. A dependent node located in between two independent nodes will be linearly coupled to these two in proportion to the distances.

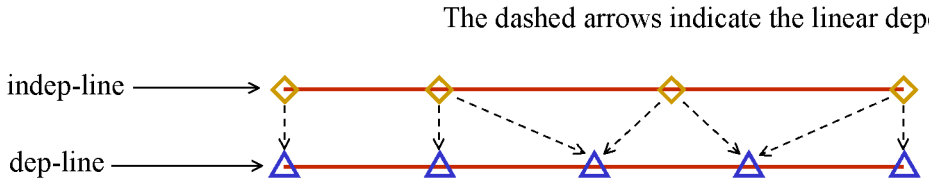

The dashed arrows indicate the linear dependency relations
indep-line
dep-line
The dep-line and indep-line must have the same location,
they are here drawn apart only for illustration purposes.

Figure 5.38 Line-line dependency

PARAMETERS:

dep-line	Select lines to be dependent. See Section 5.1 on how to perform a selection.
indep-line	Select the independent lines. See Section 5.1 on how to perform a selection.
ALL	All d.o.f.s of the dependent nodes are linearly dependent of the corresponding d.o.f.s of the independent nodes.
TRANSLATIONS-ONLY	Only the translational d.o.f.s of the dependent nodes are linearly dependent of the corresponding d.o.f.s of the independent nodes.
SELECTED	The d.o.f.s to be dependent are to be selected individually.

DEPENDENT	The d.o.f. in question of the dependent nodes is dependent of the corresponding d.o.f. of the independent nodes.
NON-DEPENDENT	The d.o.f. in question of the dependent nodes is not dependent of the corresponding d.o.f. of the independent nodes (nor of any other d.o.f.).

NOTES:

The dependent lines should be the ones with the largest number of nodes.

PROPERTY LINEAR-DEPENDENCY RIGID-BODY-DEPENDENCY

...	RIGID-BODY-DEPENDENCY	select-geometry	point	ALL		
				TRANSLATIONS-ONLY		
				SELECTED	DEPENDENT	*6
					NON-DEPENDENT	

PURPOSE:

The command defines linear dependencies between all or selected d.o.f.s of several nodes and a single node.

The dependent nodes are in this way forced to follow the displacements of a single node as a rigid body motion.

PARAMETERS:

select-geometry	Select geometry to be dependent. See Section 5.1 on how to perform a selection.
point	The name of the point of the independent node.
ALL	All d.o.f.s of the dependent nodes are linearly dependent of the corresponding d.o.f.s of the independent nodes.
TRANSLATIONS-ONLY	Only the translational d.o.f.s of the dependent nodes are linearly dependent. These translational d.o.f.s are, however, dependent on all six d.o.f.s of the independent node. I.e. the rotations of the independent node also contribute.
SELECTED	The d.o.f.s to be dependent are to be selected individually.
DEPENDENT	The d.o.f. in question of the dependent nodes is dependent of the corresponding d.o.f. of the independent nodes.
NON-DEPENDENT	The d.o.f. in question of the dependent nodes is not dependent of the corresponding d.o.f. of the independent nodes (nor of any other d.o.f.).

PROPERTY LINEAR-DEPENDENCY TWO-NODE-DEPENDENCY

...	TWO-NODE-DEPENDENCY	dep-node	...

...	indep-node1		...	indep-node2		beta
	FORCE-INTO-SUPER	indep-node1		FORCE-INTO-SUPER	indep-node2	

PURPOSE:

The command defines linear dependency between a node and two other independent nodes. The nodes are selected by referring directly to the node numbers. This specification will be lost with a DELETE MESH command. If possible, it is therefore generally better to use the TWO-POINT-DEPENDENCY option.

All d.o.f.s of the dependent node are made linearly dependent of the corresponding d.o.f.s of two independent nodes. The displacement of the dependent d.o.f.s will be:

$$r_{dep} = r_{indep1} \cdot \beta + r_{indep2} \cdot (1 - \beta)$$

where β is a dependency factor given by the user. The program will compute a default value for β based on the projection of the dependent node onto the line between the two independent nodes. See Figure 5.39. Normally, the 'two node dependency' is used when the dependent and the two independent nodes lie on a straight line.

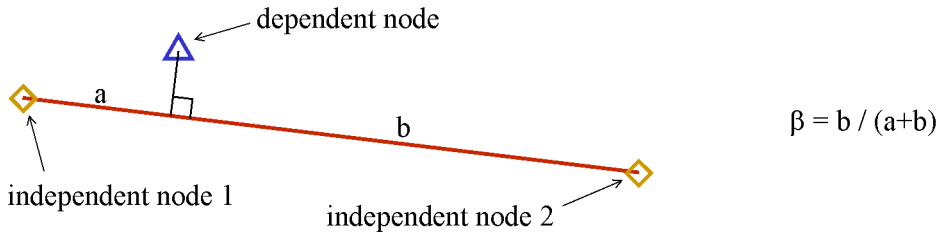


Figure 5.39 Two node linear dependency — the dependency factor β

PARAMETERS:

dep-node	Node number of the dependent node.
FORCE-INTO-SUPER	Make the d.o.f.s of the independent node super. See Section 3.5.8 for an explanation of the relevance of this option.
indep-node1	Node number of the first independent node.
indep-node2	Node number of the second independent node.
beta	Linear dependency factor.

PROPERTY LINEAR-DEPENDENCY TWO-POINT-DEPENDENCY

...	TWO-POINT-DEPENDENCY	dep-point	...

...	indep-point1	indep-node2	beta
			BY-RELATIVE-DISTANCE

PURPOSE:

The command defines linear dependencies between a node and two other independent nodes. The nodes are selected by referring to geometry points.

All d.o.f.s of the dependent node are made linearly dependent of the corresponding d.o.f.s of two independent nodes. The displacement of the dependent d.o.f.s will be:

$$r_{dep} = r_{indep1} \cdot \beta + r_{indep2} \cdot (1 - \beta)$$

where β is a dependency factor either given by the user or computed by the program based on the projection of the dependent node onto the line between the two independent nodes. See Figure 5.39. Normally, the 'two point dependency' is used when the dependent and the two independent nodes lie on a straight line.

PARAMETERS:

dep-point	Name of the dependent point.
indep-point1	Name of the first independent point.
indep-point2	Name of the second independent point.
beta	Linear dependency factor.
BY-RELATIVE-DISTANCE	The linear dependency factor is calculated by the program as explained in Figure 5.39.

PROPERTY LOAD

			BEAM-CONCENTRATED	
			COMPONENT-PRESSURE	
			CONCENTRATED	
			GRAVITY	
			HYDRO-PRESSURE	
			LINE-LOAD	
			LINE-MOMENT	
...	LOAD	load-case	NORMAL-PRESSURE	...
			PART-LINE	
			PRESCRIBED-ACCELERATION	
			PRESCRIBED-DISPLACEMENT	
			RIGID-BODY-ACCELERATION	
			RIGID-BODY-VELOCITY	
			TEMPERATURE	
			TO-MASS	

PURPOSE:

The command defines loads on the model. Loads are identified by consecutive load case numbers (1, 2, 3, ...).

A single load case may contain all and any of the load types listed above and explained in more detail in the following.

Loads refer to geometric entities (bodies, surfaces, etc.); the program will automatically transfer the loads to the FE model (the nodes and elements).

Loads are accumulated on the geometry. That is, if more than one PROPERTY LOAD command is given referring to the same load case and the same geometry then the sum of the given loads will be taken into account.

Loads may be simple real loads or complex loads consisting of real and imaginary parts. After entering the real components of a load the command END is entered if the load is to be real. By entering the imaginary components the load implicitly becomes complex.

Refer to Section 3.6 for an introduction to defining loads.

PARAMETERS:

BEAM-CONCENTRATED	Define concentrated load (force) on beam element.
COMPONENT-PRESSURE	Define component surface pressure.
CONCENTRATED	Define concentrated point load (force and moment).
GRAVITY	Define gravity load.
HYDRO-PRESSURE	Identify surfaces to be subjected to hydrostatic and hydrodynamic pressures computed by a subsequent Wadam analysis.
LINE-LOAD	Define line distributed load.
LINE-MOMENT	Define line distributed moment.
NORMAL-PRESSURE	Define normal surface pressure.
PART-LINE	Define constant line distributed load on part of a line.
PRESCRIBED-ACCELERATION	Define prescribed acceleration.
PRESCRIBED-DISPLACEMENT	Define prescribed displacement.
RIGID-BODY-ACCELERATION	Define rigid body acceleration.
RIGID-BODY-VELOCITY	Define rigid body velocity (e.g. rotation).
TEMPERATURE	Define temperature differences (initial strains).
TO-MASS	This option has no relevance. (It appears as a consequence of the CHANGE PROPERTY LOAD load-case TO-MASS command.)

PROPERTY LOAD load-case BEAM-CONCENTRATED

...	BEAM-CONCENTRATED	line	...

...	GLOBAL			...
	TRANSFORMED	trnam		
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED	
			trnam	

...	fx	fy	fz	DEG-PHASE-ANGLE	phx	phy	phz	point	distance
				IMAGINARY-COMPLEX	ifx	ify	ifz		
				RAD-PHASE-ANGLE	phx	phy	phz		
				END					

PURPOSE:

The command defines concentrated loads (forces) acting on two-node beam elements. This load differs from the CONCENTRATED alternative in that it may be applied anywhere along a line. On the other hand, it is only for two-node beam elements and it does not include moments.

Alternatively to giving concentrated loads of constant magnitude varying loads may be specified by functions; see Section 5.3.

PARAMETERS:

line	Select a single line.
GLOBAL	The load components refer to the model's cartesian coordinate system.
TRANSFORMED	The load components refer to a previously defined transformation of the cartesian coordinate system, see the command DEFINE TRANSFORMATION.
LOCAL-COORDINATE-SYSTEM	The load components refer to a previously defined (cylindrical or spherical) coordinate system, see the command DEFINE COORDINATE-SYSTEM.
UNTRANSFORMED	No additional transformation of the (cylindrical or spherical) coordinate system is performed.
trnam	Name of a previously transformation.
coord-name	Name of a previously defined coordinate system.
fx fy fz	Load (force) components.

DEG-PHASE-ANGLE	The load is complex and given as amplitudes (fx, fy and fz) and phase angles given in degrees (phx, phy and phz).
IMAGINARY-COMPLEX	The load is complex and given as real values (fx, fy and fz) and imaginary values (ifx, ify and ifz).
RAD-PHASE-ANGLE	The load is complex and given as amplitudes (fx, fy and fz) and phase angles given in radians (phx, phy and phz).
END	Entering END rather than any of the alternatives DEG-PHASE-ANGLE, IMAGINARY-COMPLEX and RAD-PHASE-ANGLE implies that the load is real.
phx phy phz	Phase angles of the complex load.
ifx ify ifz	Imaginary load (force) components of the complex load.
point	One of the two end points of the selected line, the load is applied at a given distance from this point.
distance	Distance from the selected point where the load is applied.

NOTES:

The BEAM-CONCENTRATED load can only be used for lines for which two-node beam elements are created.

The given concentrated load is changed by the program into a very short line load. The length (or width) and intensity of this line load is:

load-width = length-of-relevant-beam-element * 0.001

load-intensity = given-load / load-width

If the given distance from the selected point to the load places the load within half the load-width from the end of a beam element then the load is moved slightly so as to act on a single beam element.

PROPERTY LOAD load-case COMPONENT-PRESSURE

...	COMPONENT-PRESSURE	select-surfaces	...

...	GLOBAL			...
	TRANSFORMED	trnam		
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED	
			trnam	

...	px	py	pz	ipx	ipy	ipz	...	INSIDE-SURFACE	
								MIDDLE-SURFACE	
								OUTSIDE-SURFACE	
				END				INSIDE-LAYER	layer
								MIDDLE-LAYER	
								OUTSIDE-LAYER	

PURPOSE:

The command defines component pressure loads (pressure in x-, y- and z-component form) on surfaces.

For shell elements the pressure is applied to the inside, middle or outside surface of the elements. For the concept of inside and outside of surfaces see Section 3.12.3. The inside, middle and outside layer is similar but is relevant for layered elements only.

For solid elements the inside, middle and outside surface specification must be entered but the data is irrelevant and is not used.

Alternatively to giving constant pressure components varying pressures may be specified by functions; see Section 5.3.

PARAMETERS:

select-surfaces	Select surfaces. See Section 5.1 on how to perform a selection.
GLOBAL	The pressure components refer to the model's cartesian coordinate system.
TRANSFORMED	The pressure components refer to a previously defined transformation of the cartesian coordinate system, see the command DEFINE TRANSFORMATION.
LOCAL-COORDINATE-SYSTEM	The pressure components refer to a previously defined (cylindrical or spherical) coordinate system, see the command DEFINE COORDINATE-SYSTEM.

UNTRANSFORMED	No additional transformation of the (cylindrical or spherical) coordinate system is performed.
trnam	Name of a previously defined transformation.
coord-name	Name of a previously defined coordinate system.
px py pz	Pressure components.
ipx ipy ipz	Imaginary pressure components. By entering data for these the load will implicitly become a complex load. Entering END rather than 'ipx ipy ipz' implies that the load is real.
INSIDE-SURFACE	The pressure is applied to the inside of the surfaces.
MIDDLE-SURFACE	The pressure is applied to the middle of the surfaces.
OUTSIDE-SURFACE	The pressure is applied to the outside of the surfaces.
INSIDE-LAYER	The pressure is applied to the inside of the specified layer.
MIDDLE-LAYER	The pressure is applied to the middle of the specified layer.
OUTSIDE-LAYER	The pressure is applied to the outside of the specified layer.
layer	Layer number, see Section 3.10.2.

PROPERTY LOAD load-case CONCENTRATED

...	CONCENTRATED	select-points	...

...	GLOBAL			...
	TRANSFORMED	trnam		
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED	
			trnam	

...	fx	fy	fz	mx	my	mz	ifx	ify	ifz	imx	imy	imz
							END					

PURPOSE:

The command defines concentrated loads (forces and moments) in points. This load differs from the BEAM-CONCENTRATED alternative in that it can only be applied in geometry points.

Alternatively to giving concentrated loads of constant magnitude varying loads may be specified by functions; see Section 5.3.

PARAMETERS:

select-points	Select points. See Section 5.1 on how to perform a selection.
GLOBAL	The load components refer to the model's cartesian coordinate system.
TRANSFORMED	The load components refer to a previously defined transformation of the cartesian coordinate system, see the command DEFINE TRANSFORMATION.
LOCAL-COORDINATE-SYSTEM	The load components refer to a previously defined (cylindrical or spherical) coordinate system, see the command DEFINE COORDINATE-SYSTEM.
UNTRANSFORMED	No additional transformation of the (cylindrical or spherical) coordinate system is performed.
trnam	Name of a previously transformation.
coord-name	Name of a previously defined coordinate system.
fx fy fz mx my mz	Load (force and moment) components.
ifx ify ifz imx imy imz	Imaginary load (force and moment) components. By entering data for these the load will implicitly become a complex load.

Entering END rather than ‘ifx ify ... imz’ implies that the load is real.

NOTES:

The point selected for the concentrated load need not be a part of the geometry model for which a FE mesh is created. It may also be a ‘loose’ point which is positioned within the general coordinate tolerance (see the SET TOLERANCE COORDINATES command) from a node of the FE model. In such case the load is applied to this node. There is a risk, however, that the node created by the automatic meshing does not ‘happen to’ coincide with the point in which case the load is not applied to the model. A new program version with an updated meshing algorithm may also position the nodes differently and the load will be lost. Consequently, this feature should be used with care.

PROPERTY LOAD load-case GRAVITY

...	GRAVITY	...

...	GLOBAL		...	FLEXIBLE-PART-CONTRIBUTION	gx	gy	gz
	TRANSFORMED	trnam		STIFF-END-CONTRIBUTION			

PURPOSE:

The command defines a constant acceleration field which is used to create an inertia load, e.g. the gravitational load. The three components of the acceleration field are specified; the analysis program Sestra will compute the inertia load taking the volume of the elements and the material density into account. Point masses will also contribute to the gravity load.

PARAMETERS:

GLOBAL	The inertia load refers to the model's cartesian coordinate system.
TRANSFORMED	The inertia load refers to a previously defined transformation of the cartesian coordinate system, see the command DEFINE TRANSFORMATION.
trnam	Name of a previously defined transformation.
FLEXIBLE-PART-CONTRIBUTION	This is only relevant for beam elements and involves that if eccentricities (offsets) are employed the flexible part of the beam rather than the node-to-node part will contribute to the inertia load. For other elements types this entry is neglected.
STIFF-END-CONTRIBUTION	The node-to-node part of beam elements contribute to the inertia load, i.e. eccentricities, if present, are neglected when calculating the inertia load.
gx gy gz	The three components of the acceleration field. The default values (accepted by hitting carriage return) are gx=0, gy=0, gz=-9.81 consistent with a Z-axis pointing vertically upwards.

PROPERTY LOAD load-case HYDRO-PRESSURE

...	HYDRO-PRESSURE	select-surfaces	INSIDE	...	INSIDE-SURFACE	
					MIDDLE-SURFACE	
					OUTSIDE-SURFACE	
			OUTSIDE		INSIDE-LAYER	layer
					MIDDLE-LAYER	
					OUTSIDE-LAYER	

PURPOSE:

The command identifies surfaces that are to be subjected to hydrostatic and hydrodynamic pressures computed in a subsequent hydrodynamic analysis using Wadam. In which direction the pressures act, i.e. on which sides of the surfaces the fluid is, is decided by determining the wet surfaces by referring to the inside or outside of the surfaces.

For shell elements the HYDRO-PRESSURE is applied to the inside, middle or outside surface of the elements. For the concept of inside and outside of surfaces see Section 3.12.3. The inside, middle and outside layer is similar but is relevant for layered elements.

To understand the reason for specifying first inside/outside and then inside/middle/outside surface consider this: The former specification determines the direction of the pressure computed by Wadam (which is the wet and which is the dry side) whereas the latter specification determines where the pressure is going to act. Obviously, in normal cases INSIDE should be followed by INSIDE-SURFACE and OUTSIDE should be followed by OUTSIDE-SURFACE.

For solid elements the inside, middle and outside surface specification must be entered but the data is irrelevant and is not used.

The HYDRO-PRESSURE is used for two kinds of fluid pressures, both to be computed in subsequent Wadam analyses:

- Hydrostatic and hydrodynamic pressures on the outside, the hull, of a floating object: The identified surfaces, the wet surfaces, must extend at least up to the water line. (The exact position of the water line is determined by Wadam.) This load case number must be 1.

- Fluid pressures in tanks: The identified surfaces, the inside of tanks, must extend up to the fluid level, i.e. the surfaces identifies the level of the fluid. The load case number must be 2 or higher. This number is at the same time an identification of tank number which is referred to in the Wadam (or rather Prewad) input when defining tank properties.

The HYDRO-PRESSURE load case(s) do not in themselves define any loading conditions. When combining loads in Presel (in connection with a superelement analysis) and interpreting results from the hydrodynamic and structural analyses, these HYDRO-PRESSURE load cases are irrelevant.

PARAMETERS:

select-surfaces	Select surfaces. See Section 5.1 on how to perform a selection.
INSIDE	The inside is the wet surface.
OUTSIDE	The outside is the wet surface.
INSIDE-SURFACE	Pressure is acting on the inside of the surfaces.
OUTSIDE-SURFACE	Pressure is acting on the outside of the surfaces.
MIDDLE-SURFACE	Pressure is acting in the middle-surface.
INSIDE-LAYER	The pressure is applied to the inside of the specified layer.
MIDDLE-LAYER	The pressure is applied to the middle of the specified layer.
OUTSIDE-LAYER	The pressure is applied to the outside of the specified layer.
layer	Layer number, see Section 3.10.2.

NOTES:

This load case shall be applied both to the panel model for hydrodynamic analysis and the FE model for structural analysis.

PROPERTY LOAD load-case LINE-LOAD

...	LINE-LOAD	select-lines	...

...	GLOBAL			...
	TRANSFORMED	trnam		
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED	
			trnam	

...	fx	fy	fz	ifx	ify	ifz	...	INSIDE-SURFACE-SHELL-ELEMENT
								MIDDLE-SURFACE-SHELL-ELEMENT
								OUTSIDE-SURFACE-SHELL-ELEMENT
				END				BEAM-ELEMENTS
								MEMBRANE-ELEMENT
								MIDDLE-LAYER-LAYERED-ELEMENT layer

PURPOSE:

The command defines line-distributed loads (forces) acting on lines.

The line load must be defined to be acting on beam elements, membrane elements or shell elements. If the load is defined to be acting on, say a beam element but only shell elements are present, then no load will apply! If e.g. both beam and shell elements are present then the load may be applied to either of them (assuming the inside/outside of the shell elements and beam eccentricities have no consequence). Concerning line load on membrane elements see the warning under NOTES below.

For shell elements the load is applied to the inside, middle or outside surface of the elements. For the concept of inside and outside of surfaces see Section 3.12.3. For layered elements the load is applied to the middle layer of an identified layer number.

Alternatively to giving constant load components varying loads may be specified by functions; see Section 5.3.

PARAMETERS:

select-lines	Select lines. See Section 5.1 on how to perform a selection.
GLOBAL	The load components refer to the model's cartesian coordinate system.
TRANSFORMED	The load components refer to a previously defined transformation of the cartesian coordinate system, see the command DEFINE TRANSFORMATION.

LOCAL-COORDINATE-SYSTEM	The load components refer to a previously defined (cylindrical or spherical) coordinate system, see the command DEFINE COORDINATE-SYSTEM.
UNTRANSFORMED	No additional transformation of the (cylindrical or spherical) coordinate system is performed.
trnam	Name of a previously defined transformation.
coord-name	Name of a previously defined coordinate system.
fx fy fz	Load components.
ifx ify ifz	Imaginary load components. By entering data for these the load will implicitly become a complex load. Entering END rather than 'ifx ify ifz' implies that the load is real.
INSIDE-SURFACE-SHELL-ELEMENT	The load applies to the inside of shell surfaces.
MIDDLE-SURFACE-SHELL-ELEMENT	The load applies to the middle of shell surfaces.
OUTSIDE-SURFACE-SHELL-ELEMENT	The load applies to the outside of shell surfaces.
BEAM-ELEMENTS	The load applies to beam elements.
MEMBRANE-ELEMENT	The load applies to membrane elements.
MIDDLE-LAYER-LAYERED-ELEMENT	The load applies to the specified layer of layered elements.
layer	Layer number, see Section 3.10.2.

NOTES:

The membrane element has by definition no stiffness perpendicular to its surface, the element can only carry in-plane loads. When a line load is applied to membrane elements Prefem will, therefore, discard any components out-of-plane as shown in the example of Figure 5.40.

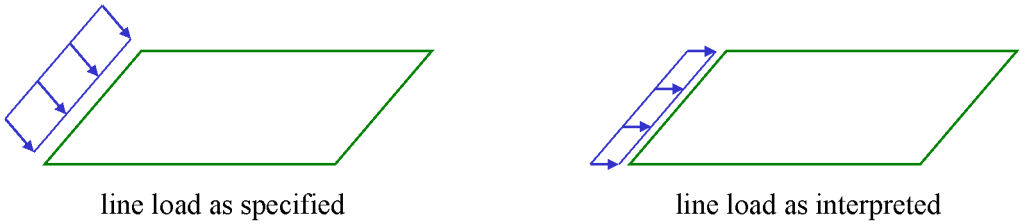


Figure 5.40 Line load components out-of-plane for membrane elements are neglected

Now consider the example of Figure 5.41. The vertical plus horizontal membrane elements are physically able to carry the line load illustrated in the left-most sketch. However, Prefem will apply such a load to either the horizontal or the vertical membrane, which of them cannot easily be foreseen by the user. And in either case the horizontal or the vertical component of the load will be discarded as illustrated by the two

sketches in the middle. The user, therefore, has to decompose the line load and apply each component separately as illustrated by the right-most sketch.

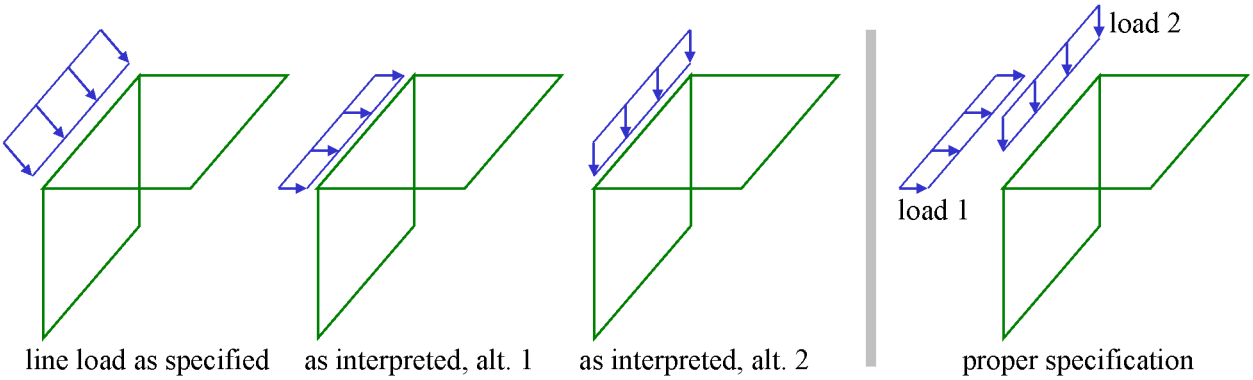


Figure 5.41 Incorrect and correct specification of line loads for membrane model

PROPERTY LOAD load-case LINE-MOMENT

...	LINE-MOMENT	select-lines	...

...	GLOBAL			...
	TRANSFORMED	trnam		
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED	
			trnam	

...	mx	my	mz	imx	imy	imz	...	INSIDE-SURFACE-SHELL-ELEMENT
								MIDDLE-SURFACE-SHELL-ELEMENT
								OUTSIDE-SURFACE-SHELL-ELEMENT
				END				BEAM-ELEMENTS
								MEMBRANE-ELEMENT
								MIDDLE-LAYER-LAYERED-ELEMENT layer

PURPOSE:

The command defines line-distributed moments acting on lines.

The line moment must be defined to be acting on either three node beam elements or shell elements. If the moment is defined to be acting on, say a beam element but only shell elements are present, then no load will apply! If e.g. both beam and shell elements are present then the moment may be applied to either of them.

For shell elements the moment is applied to the inside, middle or outside surface of the elements. For the concept of inside and outside of surfaces see Section 3.12.3. For layered elements the moment is applied to the middle layer of an identified layer number.

Alternatively to giving constant moment components varying moments may be specified by functions; see Section 5.3.

PARAMETERS:

See the explanation for the line load.

PROPERTY LOAD load-case NORMAL-PRESSURE

...	NORMAL-PRESSURE	select-surfaces	p	ip	...	INSIDE-SURFACE	
						MIDDLE-SURFACE	
						OUTSIDE-SURFACE	
				END		INSIDE-LAYER	layer
						MIDDLE-LAYER	
						OUTSIDE-LAYER	

PURPOSE:

The command defines normal pressure loads, e.g. hydrostatic and air pressures, on surfaces.

For shell elements the pressure is applied to the inside, middle or outside surface of the elements. For the concept of inside and outside of surfaces see Section 3.12.3. The inside, middle and outside layer is similar but is relevant for layered elements.

For solid elements the inside, middle and outside surface specification is irrelevant and not used.

As opposed to the component pressure for which the direction of the pressure is solely determined by the sign of the pressure components the direction of the normal pressure is determined by its sign combined with the inside/outside definition. See Section 3.12.3 for an explanation of this.

Alternatively to giving a constant normal pressure a varying pressure may be specified by functions; see Section 5.3.

PARAMETERS:

select-surfaces	Select surfaces. See Section 5.1 on how to perform a selection.
p	Normal pressure. A positive pressure points from the outside towards the inside.
ip	Imaginary normal pressure. Entering END implies that the load is real.
INSIDE-SURFACE	The pressure is applied to the inside of the surfaces.
MIDDLE-SURFACE	The pressure is applied to the middle of the surfaces.
OUTSIDE-SURFACE	The pressure is applied to the outside of the surfaces.
INSIDE-LAYER	The pressure is applied to the inside of the specified layer.
MIDDLE-LAYER	The pressure is applied to the middle of the specified layer.
OUTSIDE-LAYER	The pressure is applied to the outside of the specified layer.
layer	Layer number, see Section 3.10.2.

PROPERTY LOAD load-case PART-LINE

...	PART-LINE	line	...

...	GLOBAL			...
	TRANSFORMED	trnam		
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED	
			trnam	

...	fx	fy	fz	DEG-PHASE-ANGLE	phx	phy	phz	point1	dist1	point2	dist2	...
				IMAGINARY-COMPLEX	ifx	ify	ifz					
				RAD-PHASE-ANGLE	phx	phy	phz					
				END								

...	INSIDE-SURFACE-SHELL-ELEMENT	
	MIDDLE-SURFACE-SHELL-ELEMENT	
	OUTSIDE-SURFACE-SHELL-ELEMENT	
	BEAM-ELEMENTS	
	MEMBRANE-ELEMENT	
	MIDDLE-LAYER-LAYERED-ELEMENT	layer

PURPOSE:

The command defines partially distributed line-loads (forces) of constant magnitude acting on lines.

The part-line load must be defined to be acting on either beam elements, membrane elements or shell elements. If the load is defined to be acting on, say a beam element but only shell elements are present, then no load will apply! If e.g. both beam and shell elements are present, then the load may be applied to either of them. The choice between beam and shell elements has consequences though, see under NOTES below. Concerning part-line load on membrane elements, see the warning under NOTES of the command PROPERTY LOAD load-case LINE-LOAD.

For shell elements the load is applied to the inside, middle or outside surface of the elements. For the concept of inside and outside of surfaces, see Section 3.12.3. For layered elements the load is applied to the middle layer of an identified layer number.

PARAMETERS:

line	Select a single line.
GLOBAL	The load components refer to the model's cartesian coordinate system.

TRANSFORMED	The load components refer to a previously defined transformation of the cartesian coordinate system, see the command DEFINE TRANSFORMATION.
LOCAL-COORDINATE-SYSTEM	The load components refer to a previously defined (cylindrical or spherical) coordinate system, see the command DEFINE COORDINATE-SYSTEM.
UNTRANSFORMED	No additional transformation of the (cylindrical or spherical) coordinate system is performed.
trnam	Name of a previously defined transformation.
coord-name	Name of a previously defined coordinate system.
fx fy fz	Constant load (force) components, i.e. functions cannot be given.
DEG-PHASE-ANGLE	The load is complex and given as amplitudes (fx, fy and fz) and phase angles given in degrees (phx, phy and phz).
IMAGINARY-COMPLEX	The load is complex and given as real values (fx, fy and fz) and imaginary values (ifx, ify and ifz).
RAD-PHASE-ANGLE	The load is complex and given as amplitudes (fx, fy and fz) and phase angles given in radians (phx, phy and phz).
END	Entering END rather than any of the alternatives DEG-PHASE-ANGLE, IMAGINARY-COMPLEX and RAD-PHASE-ANGLE implies that the load is real.
phx phy phz	Phase angles of the complex load (in degrees or radians depending on the chosen option).
ifx ify ifz	Imaginary load (force) components.
point1	One of the two end points of the selected line, the load starts at a given distance from this point.
dist1	Distance from the first selected point where the load starts.
point2	The other of the two end points of the selected line, the load ends at a given distance from this point.
dist2	Distance from the second selected point where the load ends.
INSIDE-SURFACE-SHELL-ELEMENT	The load applies to the inside of shell surfaces.
MIDDLE-SURFACE-SHELL-ELEMENT	The load applies to the middle of shell surfaces.
OUTSIDE-SURFACE-SHELL-ELEMENT	The load applies to the outside of shell surfaces.
BEAM-ELEMENTS	The load applies to beam elements.

MEMBRANE-ELEMENT	The load applies to membrane elements.
MIDDLE-LAYER-LAYERED-ELEMENT	The load applies to the specified layer of layered elements.
layer	Layer number, see Section 3.10.2.

NOTES:

Only the two-node beam element can handle a line load on part of its length. Three-node beam, shell and membrane elements cannot handle line loads on part of their lengths/edges. This is accounted for as follows: The load is automatically distributed to elements within the specified load length. If the start or end position of the load is inside an element then the extent of the load is automatically adjusted to match whole elements. In such cases the load value is scaled to compensate for the adjustment.

See the notes of the LINE-LOAD type of load for additional information on load applied to membrane elements.

PROPERTY LOAD load-case PRESCRIBED-ACCELERATION

...	PRESCRIBED-ACCELERATION	select-geometry	...

...	GLOBAL			...
	TRANSFORMED	trnam		
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED	
			trnam	

...	ax	ay	az	arx	ary	arz	iax	iay	iaz	iarx	iary	iarz
							END					

PURPOSE:

The command defines prescribed accelerations. This type of load is only relevant for a subsequent dynamic analysis (in Sestra). A typical application may be to give the foundation of a structure an acceleration and do a dynamic analysis.

Note: As opposed to the gravity load this command does not define an acceleration field.

The geometry subjected to the accelerations must be given the boundary condition PRESCRIBED-ACCELERATION using the PROPERTY BOUNDARY command.

PARAMETERS:

select-geometry	Select geometry. See Section 5.1 on how to perform a selection.
GLOBAL	The acceleration components refer to the model's cartesian coordinate system.
TRANSFORMED	The acceleration components refer to a previously defined transformation of the cartesian coordinate system, see the command DEFINE TRANSFORMATION.
LOCAL-COORDINATE-SYSTEM	The acceleration components refer to a previously defined (cylindrical or spherical) coordinate system, see the command DEFINE COORDINATE-SYSTEM.
UNTRANSFORMED	No additional transformation of the (cylindrical or spherical) coordinate system is performed.
trnam	Name of a previously defined transformation.
coord-name	Name of a previously defined coordinate system.
ax ay az arx ary arz	Acceleration components.

iax iay iaz iarx iary iarz	Imaginary acceleration components. By entering data for these the acceleration field will implicitly become complex. Entering END rather than ‘iax iay ... iarz’ implies that the acceleration field is real.

PROPERTY LOAD load-case PRESCRIBED-DISPLACEMENT

...	PRESCRIBED-DISPLACEMENT	select-geometry	...

...	GLOBAL			...
	TRANSFORMED	trnam		
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED	
			trnam	

...	dx	dy	dz	drx	dry	drz	idx	idy	idz	idrx	idry	idrz
							END					

PURPOSE:

The command defines prescribed displacements.

The geometry subjected to the displacements must be given the boundary condition PRESCRIBED-DISPLACEMENT using the PROPERTY BOUNDARY command.

PARAMETERS:

select-geometry	Select geometry. See Section 5.1 on how to perform a selection.
GLOBAL	The displacement components refer to the model's cartesian coordinate system.
TRANSFORMED	The displacement components refer to a previously defined transformation of the cartesian coordinate system, see the command DEFINE TRANSFORMATION.
LOCAL-COORDINATE-SYSTEM	The displacement components refer to a previously defined (cylindrical or spherical) coordinate system, see the command DEFINE COORDINATE-SYSTEM.
UNTRANSFORMED	No additional transformation of the (cylindrical or spherical) coordinate system is performed.
trnam	Name of a previously defined transformation.
coord-name	Name of a previously defined coordinate system.
dx dy dz drx dry drz	Displacement components, rotations are given in radians.
idx idy idz idrx idry idrz	Imaginary displacement components. By entering data for these the displacement field will implicitly become complex. Entering END rather than 'idx idy ... idrz' implies that the displacement field is real.

PROPERTY LOAD load-case RIGID-BODY-ACCELERATION

...	RIGID-BODY-ACCELERATION	GLOBAL	...

...	ax	ay	az	arx	ary	arz	iax	iay	iaz	iarx	iary	iarz
							END					

PURPOSE:

The command defines an acceleration field. The effect of this command may be compared with that of the GRAVITY load (see the notes below) but in addition to translational components the present command allows specification of rotational components. The analysis program Sestra will compute the inertia load taking the volume of the elements and the material density into account. Point masses will also contribute to the gravity load.

The rotational acceleration components refer to the axes of the cartesian coordinate system of the model.

PARAMETERS:

GLOBAL — The cartesian coordinate system of the model is used.

ax ay az arx ary arz — The acceleration components. The unit of the rotational acceleration is radians/second^2.

iax iay iaz iarx iary iarz — Imaginary acceleration components. Entering END implies that the load is real.

NOTES:

The following example is included as an illustration of the effect of the RIGID-BODY-ACCELERATION load type compared with the GRAVITY load type.

The accelerations given for the RIGID-BODY-ACCELERATION are in effect applied to the fixed nodes of the model. Therefore, the following two loads will produce the same results as illustrated for a simple cantilever beam in Figure 5.42:

```
PROPERTY LOAD 1 GRAVITY GLOBAL FLEXIBLE-PART-CONTRIBUTION 0.0 0.0 -9.81
..
PROPERTY LOAD 2 RIGID-BODY-ACCELERATION GLOBAL 0.0 0.0 9.81 0.0 0.0 0.0 END
..
```

Figure 5.42 RIGID-BODY-ACCELERATION compared with GRAVITY

PROPERTY LOAD load-case RIGID-BODY-VELOCITY

...	RIGID-BODY-VELOCITY	GLOBAL	...

...	vx	vy	vz	vrx	vry	vrz	ivx	ivy	ivz	ivrx	ivry	ivrz
							END					

PURPOSE:

The command defines a velocity field.

The rotational velocity components refer to the axes of the cartesian coordinate system of the model.

This feature is presently not available in the analysis program Sestra.

PARAMETERS:

GLOBAL	The cartesian coordinate system of the model is used.
vx vy vz vrx vry vrz	The velocity components. The unit of the rotational velocity is radians/second.
ivx ivy ivz ivrx ivry ivrz	Imaginary velocity components. Entering END implies that the load is real.

PROPERTY LOAD load-case TEMPERATURE

...	TEMPERATURE	select-geometry	ONE-VALUE-EACH-NODE	temp-diff	
			TWO-VALUES-ON-SHELL	temp-diff1	temp-diff2
			VALUE-ON-NEIGHBOURING-ELEMENTS		temp-diff

PURPOSE:

The command defines initial strains due to thermal expansion by a given temperature difference between the present temperature and a reference temperature. That is, the values to give are the temperature changes compared to an initial and strain less state. The thermal expansion coefficient is defined by the PROPERTY MATERIAL command.

The temperature difference is assigned to all elements within the selected geometry. Repetitive use of this command does not involve accumulation of temperatures like other load types, new values will replace previous values. Care must be taken when assigning temperature differences to avoid giving the same node of an element different values as there is no way of determining which value will be used. If two or more different values are assigned to a node given in two different load commands, any of the given values may be assigned to the element.

The three methods of assigning temperature loads have one significant difference: ONE-VALUE-EACH-NODE and TWO-VALUES-ON-SHELL assign the temperature only to the elements of the selected geometry itself, whereas VALUE-ON-NEIGHBOURING-ELEMENTS also assigns the temperature to elements adjoining the selected geometry. For example, using the VALUE-ON-NEIGHBOURING-ELEMENTS option to give a temperature for a selected surface will also assign the temperature to the nodes, in the selected surface, of adjoining solid elements. However, elements on the borderlines of the surface will not be assigned the temperature and neither will elements of neighbouring surfaces.

Another example: Using the VALUE-ON-NEIGHBOURING-ELEMENTS option to give a temperature for a selected line will also assign the temperature to the nodes, on the selected line, of adjoining shell and solid elements.

Yet another example: Using the VALUE-ON-NEIGHBOURING-ELEMENTS option to give a temperature for a selected body will only assign the temperature to the solid elements of that body.

PARAMETERS:

select-geometry

Select geometry. Points cannot be selected. See Section 5.1 on how to perform a selection.

ONE-VALUE-EACH-NODE

One value of the temperature difference is assigned to each node. Can be used for any element type.

TWO-VALUES-ON-SHELL

Two values of the temperature difference are assigned to each node. This option can be used for shell elements only. The values given are the tem-

	perature differences on the inside and outside of the surface, the inside given first.
VALUE-ON-NEIGHBOURING-ELEMENTS	See the explanation above.
temp-diff	Temperature difference
tempdiff1 tempdiff2	Temperature differences for the inside and outside surface of shell elements, respectively

NOTES:

Repetitive specification of temperature differences will override previous values.

For twenty node solid elements the mid-side nodes will get the average of the corresponding corner nodes.

PROPERTY LOCAL-COORDINATE-BEAM

...	LOCAL-COORDINATE-BEAM	YX-PLANE	...
		ZX-PLANE	

...	+X-GLOBAL-INFINITY				select-lines
	-X-GLOBAL-INFINITY				
	+Y-GLOBAL-INFINITY				
	-Y-GLOBAL-INFINITY				
	+Z-GLOBAL-INFINITY				
	-Z-GLOBAL-INFINITY				
	GUIDING-POINT-DIRECTION	x	y	z	
	FROM-FIXED-POINT				
	TOWARDS-FIXED-POINT				

PURPOSE:

The command defines local coordinate systems for beams elements. A new definition of local coordinate systems for the same element will override the former definition. Local coordinate systems cannot be deleted. For beam elements for which no local coordinate system has been defined a default condition takes effect as explained below.

The element local x-axis is by definition the neutral axis of the cross section and points from the first node towards the second node. The first and second nodes are implicitly defined by the command defining the lines. The local y-z-plane is normal to the local x-axis. Defining a local coordinate system involves determining the orientation of either of the local y- and z-axes. The right-hand-rule determines the third local axis.

The orientation of the local y- and z-axes is defined by either determining the orientation of y-x-plane or the z-x-plane. These planes are given an orientation by use of a guiding point. This guiding point may be defined by:

- Specifying its location relative to end 1 of the element (GUIDING-POINT-DIRECTION)
- Giving its coordinates directly (FROM-FIXED-POINT and TOWARDS-FIXED-POINT)
- Positioning it infinitely far away along any of the global axes, either in positive or in negative direction (the ...GLOBAL-INFINITY options)

The guiding point will lie in the y-x- or z-x-plane on the positive y- or z-side respectively. This is, however, not true for the FROM-FIXED-POINT alternative which involves that the guiding point lies on the negative y/z-side.

For two node beam elements the local coordinate system is constant along the local x-axis. For three node beam elements the local coordinate system may differ between the three nodes. The local x-axis will be tangential to the parabola through the three nodes.

Beam elements for which local coordinate systems are not explicitly defined will be given default local coordinate systems as follows: The local z-x-plane is parallel with the global Z-axis and with the positive direction of the local z-axis in the direction of the positive global Z-axis (as if the command PROPERTY LOCAL-COORDINATE-BEAM ZX-PLANE +Z-GLOBAL-INFINITY was given). If the local x-axis is parallel with the global Z-axis then the local z-axis is defined to be parallel with the global Y-axis.

The local coordinate systems of the elements can be tabulated by the PRINT ELEMENT select-element LOCAL-COORDINATE command. The right-hand column of this print table indicates whether the local coordinate system has been explicitly defined (SPEC.) or calculated by the default condition described above (CALC.).

PARAMETERS:

YX-PLANE	The local y-x-plane is to be defined.
ZX-PLANE	The local z-x-plane is to be defined.
+X-GLOBAL-INFINITY	The guiding point is positioned infinitely far out along the positive global X-axis.
-X-GLOBAL-INFINITY	The guiding point is positioned infinitely far out along the negative global X-axis.
+Y-GLOBAL-INFINITY	The guiding point is positioned infinitely far out along the positive global Y-axis.
-Y-GLOBAL-INFINITY	The guiding point is positioned infinitely far out along the negative global Y-axis.
+Z-GLOBAL-INFINITY	The guiding point is positioned infinitely far out along the positive global Z-axis.
-Z-GLOBAL-INFINITY	The guiding point is positioned infinitely far out along the negative global Z-axis.
GUIDING-POINT-DIRECTION	The guiding point is positioned at a given vector away from end 1 of the element.
FROM-FIXED-POINT	The guiding point is a fixed point and the positive direction of the local y- or z-axis, whichever is relevant, is from the fixed point and towards end 1 of the element. In contrast with the other alternatives this option, therefore, involves that the guiding point is positioned on the negative y/z-side.
TOWARDS-FIXED-POINT	The guiding point is a fixed point and the positive direction of the local y- or z-axis, whichever is relevant, is from end 1 of the element and towards the fixed point.

x y z	If GUIDING-POINT-DIRECTION: Vector given in the cartesian coordinate system and pointing from end 1 of the element towards the guiding point. If FROM/TOWARDS-FIXED-POINT: Cartesian coordinates of the guiding point.
select-lines	Select lines. See Section 5.1 on how to perform a selection.

NOTES:

If a local coordinate system has been defined and either of the element nodes are repositioned then the local coordinate system may become erroneous (the local x-axis may change while the local y- and z-axes are fixed). Introducing eccentricities may also cause this. In such cases the local coordinate system must be redefined by the user.

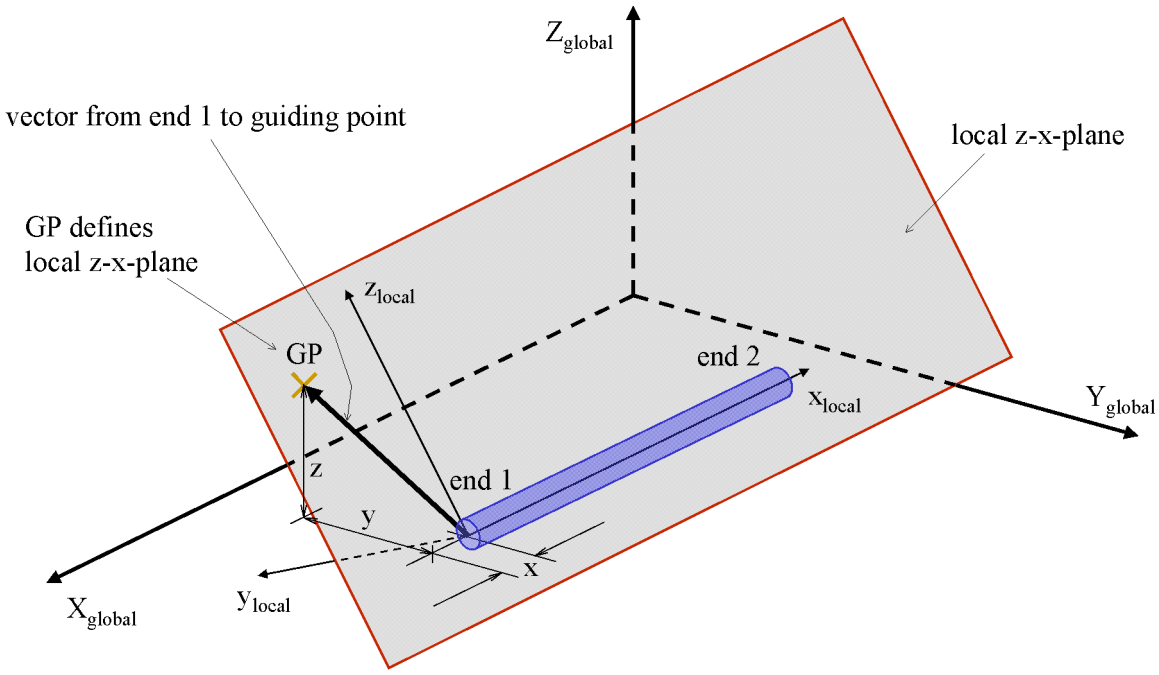


Figure 5.43 The local coordinate system defined by a guiding point

PROPERTY LOCAL-COORDINATE-SURFACE

...	LOCAL-COORDINATE-SURFACE	select-surfaces	...

...	GLOBAL			...
	TRANSFORMED	trnam		
	LOCAL-COORDINATE-SYSTEM	coord-name	UNTRANSFORMED	
			trnam	

PURPOSE:

The command defines local coordinate systems *for layered elements only*. It has no effect for other elements.

The local coordinate system is used to give orientation to stiffeners of the layered element. See the DEFINE LAYERED command, also see Section 3.10.2.

PARAMETERS:

select-surfaces — Select surfaces. See Section 5.1 on how to perform a selection.

GLOBAL — The local x-axis of the element is defined by the projection onto the element of the global X-axis.

If the global X-axis is normal to the element then the local x-axis of the element will be parallel with the global Y-axis.

The GLOBAL option ensures that a layered element will have the same local coordinate system as a shell element (for which the local coordinate system is calculated by Sestra).

TRANSFORMED — The local x-axis of the element is defined by the projection onto the element of the X-axis of a transformed coordinate system. This transformed coordinate system must previously have been defined by the DEFINE TRANSFORMATION command.

If the transformed X-axis is normal to the element then the local y-axis of the element will be parallel with the transformed Y-axis. Note that this solution to situations where the X-axis is normal to the element plane differs from that of the GLOBAL option. This means that even when the transformation matrix equals the identity matrix the TRANSFORMED and GLOBAL options will give different results in certain situations.

This option ensures that neighbouring layered elements in most cases will have smooth transitions in local coordinates, even in cases where the normal shell elements will have discontinuity in their local coordinate systems. If this behaviour is preferred

	you may use this option with the identity matrix as dummy transformation.
LOCAL-COORDINATE-SYSTEM	The local x-axis of the element is defined by the projection onto the element of the R-axis of a cylindrical or spherical coordinate system. Such coordinate systems must previously have been defined by the DEFINE COORDINATE-SYSTEM command. The cylindrical/spherical coordinate system may even be given a transformation by referring to a previously defined transformation. If the R-axis is normal to the element then the local y-axis of the element will be parallel with the Φ-axis of the cylindrical/ spherical coordinate-system.
UNTRANSFORMED	No additional transformation of the (cylindrical or spherical) coordinate system is performed.
coord-name	Name of a previously defined coordinate system.
trnam	Name of a previously defined transformation.

NOTES:

If local coordinate system of a surface is undefined (the PROPERTY LOCAL-COORDINATE-SURFACE command has not been used) then the GLOBAL option will take effect.

This command is irrelevant for all elements but the layered element.

PROPERTY MATERIAL

...	MATERIAL	material-name	ANISOTROPIC	...
			CONTACT[a]	
			DAMPER	
			ELASTIC	
			MASS	
			NON-LINEAR-ELASTO-PLASTIC[b]	
			SPRING	

a. This option is presently inactive.
b. This option is presently inactive.

PURPOSE:

The command defines:

- Material types to be connected to elements using the CONNECT command
- Stiffness matrices of spring elements to be referred to in the DEFINE SPRING command
- Damping matrices of damper elements to be referred to in the DEFINE DAMPER command
- Mass matrices of mass elements to be referred to in the DEFINE MASS-ELEMENT command

PARAMETERS:

material-name	User-given name for the material.
ANISOTROPIC	Define an anisotropic material.
DAMPER	Define the damping coefficients for a damper element.
ELASTIC	Define an elastic material.
MASS	Define the mass matrix for a mass element.
SPRING	Define the stiffness of a spring element.

PROPERTY MATERIAL material-name ANISOTROPIC

...	ANISOTROPIC	2D-ELEMENT	...
		3D-SHELL-ELEMENT	
		SOLID-ELEMENT	

PURPOSE:

The command defines data for an anisotropic material. An orthotropic material, being a special case of the anisotropic material, may also be given.

PARAMETERS:

2D-ELEMENT	Anisotropic material is to be specified for lower order (3 and 4 node) shell elements or for membrane elements.
3D-SHELL-ELEMENT	Anisotropic material is to be specified for higher order (6 and 8 node) shell elements.
SOLID-ELEMENT	Anisotropic material is to be specified for solid elements.

NOTES:

The appropriate type of element must have been set before being allowed to connect the anisotropic materials 2D-ELEMENT and 3D-ELEMENT to surfaces.

For a more detailed description of anisotropic and orthotropic materials see Ref. /4/.

PROPERTY MATERIAL material-name ANISOTROPIC 2D-ELEMENT

...	2D-ELEMENT	q1	q2	q3	rho	...

...	d11	d21	d22	d31	d32	d33	ps1	ps2	damp1	damp2	alpha1	alpha2

PURPOSE:

The command specifies the data for either an anisotropic or orthotropic material for lower order (3 and 4 node) shell elements and membrane elements.

The anisotropic material has two material axes both lying in the element plane. The first material axis is determined by projecting a vector Q onto the element plane. The second material axis is perpendicular to the first material axis and lies in the element plane.

PARAMETERS:

q1 q2 q3 — Components of a vector Q in the cartesian coordinate system of the model determining by its projection onto the element plane the first material axis of the anisotropic material. Note that the vector Q cannot be perpendicular to any of the elements.

rho — Material density

d11 ... d33 — Terms of the lower triangular part of the general anisotropic elasticity matrix; see Equation (5.1).

ps1 ps2 — Factors to produce the stress perpendicular to the membrane plane for plane-strain problems. (For a plane-stress material, which is the normal case, these parameters are equal to zero.) The stress normal to the membrane plane will be calculated as follows:

$$\sigma_n = ps1 \cdot \sigma_1 + ps2 \cdot \sigma_2$$

For an isotropic plane-strain material ps1 and ps2 are equal to Poisson's ratio.

damp1 damp2 — Specific damping coefficients for the first and second material axes of anisotropy.

alpha1 alpha2 — Thermal expansion coefficients for the first and second material axes of anisotropy.

$$\begin{bmatrix} d11 & & \\ d21 & d22 & \\ d31 & d32 & d33 \end{bmatrix}$$

The lower triangular general anisotropic elasticity matrix (5.1)

NOTES:

An orthotropic material is defined by taking advantage of the fact that it is a special case of the anisotropic material. To define an orthotropic material in its principal axes the lower triangular part of the general anisotropic elasticity matrix should have zero values for some of the terms:

$$\begin{bmatrix} d11 & & \\ d21 & d22 & \\ 0 & 0 & d33 \end{bmatrix} \tag{5.2}$$

The elasticity matrix of an orthotropic material expressed in terms of the Young's modulus, the shear modulus and Poisson's ratio is given below.

The stress-strain relationship $\sigma = \mathbf{D} \cdot \varepsilon$ is:

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{bmatrix} = \frac{1}{1 - \nu_{12}\nu_{21}} \begin{bmatrix} E_1 & \nu_{21}E_1 & 0 \\ \nu_{12}E_2 & E_2 & 0 \\ 0 & 0 & G_{12}(1 - \nu_{12}\nu_{21}) \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{bmatrix} \tag{5.3}$$

where:

- ε_i, σ_i and E_i are the strain, stress and Young's modulus, respectively, in the direction of axis no. i.
- ν_{ij} is the coefficient expressing the negative strain in the direction of axis no. j caused by a positive strain in the direction of axis no. i ($\varepsilon_j = -\nu_{ij}\varepsilon_i$ for $\sigma_i = \sigma$ and all other stresses equal to zero).
- γ_{12} (= γ_{21}), τ_{12} (= τ_{21}) and G_{12} (= G_{21}) are the shear strain, shear stress and shear modulus, respectively, in the plane defined by axes nos. 1 and 2.

Hence, the terms of the elasticity matrix becomes:

d11 = $E_1 / (1-\nu_{12}\nu_{21})$

d21 = $\nu_{21} E_1 / (1-\nu_{12}\nu_{21}) = \nu_{12} E_2 / (1-\nu_{12}\nu_{21})$

d22 = $E_2 / (1-\nu_{12}\nu_{21})$

d31 = d32 = 0

d33 = G_{12}

The material constants Young's modulus (E) and Poisson's ratio (ν) in the two directions are related as follows:

$\nu_{21} E_1 = \nu_{12} E_2$

PROPERTY MATERIAL material-name ANISOTROPIC 3D-SHELL-ELEMENT

...	3D-SHELL-ELEMENT	q1	q2	q3	rho	nlay	...	

...	{	thick	angle	d11	d21	d22	d31	d32	d33	d41	d42	d43	d44	...

...	d51	d52	d53	d54	d55	damp1	damp2	alpha1	alpha2	}*nlay

PURPOSE:

The command specifies the data for either an anisotropic or orthotropic material for higher order (6 and 8 node) shell elements and for sandwich elements (6 and 8 node).

The anisotropic material properties are specified in a material coordinate system. The first axis of the material coordinate system is determined by projecting a vector Q onto the element plane and optionally rotate this axis a given angle around the normal to the element plane. The second material axis lies in the element plane and is perpendicular to the first axis. The third material axis is normal to the element plane.

The sandwich element, a multilayered shell element comprised of normally three but in principle any number of layers through the shell thickness, is defined by first specifying number of layers and thereafter giving thickness plus anisotropic material properties for each layer.

PARAMETERS:

q1 q2 q3 — Components of a vector Q in the cartesian coordinate system of the model determining by its projection onto the element plane the first material axis of the anisotropic material. Note that the vector Q cannot be perpendicular to any of the elements.

rho — Material density.

nlay — Number of layers of a multilayered (sandwich) material. If nlay=1 then the element reduces to an ordinary shell element with anisotropic material properties.

thick — Thickness of the layer in percent of the total element thickness (as defined by the PROPERTY THICKNESS command).

If nlay=1 then this entry is skipped and the layer constitutes 100 % of the total thickness.

For the last of several layers this entry is skipped as the thickness of the last layer will be equal to the remainder of the total thickness.

angle — Angle in degrees from the projection of the vector Q to the first material axis of anisotropy for this layer. Positive angle is positive rotation about local z-axis.

d11 ... d55 — Terms of the lower triangular part of the general anisotropic elasticity matrix; see Equation (5.4).

damp1 damp2 Specific damping coefficients for the first and second material axes of anisotropy.

alpha1 alpha2 Thermal expansion coefficients for the first and second material axes of anisotropy.

$$\begin{bmatrix} d11 & & & & \\ d21 & d22 & & & \\ d31 & d32 & d33 & & \\ d41 & d42 & d43 & d44 & \\ d51 & d52 & d53 & d54 & d55 \end{bmatrix}$$ The lower triangular general anisotropic elasticity matrix (5.4)

NOTES:

An orthotropic material is defined by taking advantage of the fact that it is a special case of the anisotropic material. To define an orthotropic material in its principal axes the lower triangular part of the general anisotropic elasticity matrix should have zero values for some of the terms:

$$\begin{bmatrix} d11 & & & & \\ d21 & d22 & & & \\ 0 & 0 & d33 & & \\ 0 & 0 & 0 & d44 & \\ 0 & 0 & 0 & 0 & d55 \end{bmatrix} \quad (5.5)$$

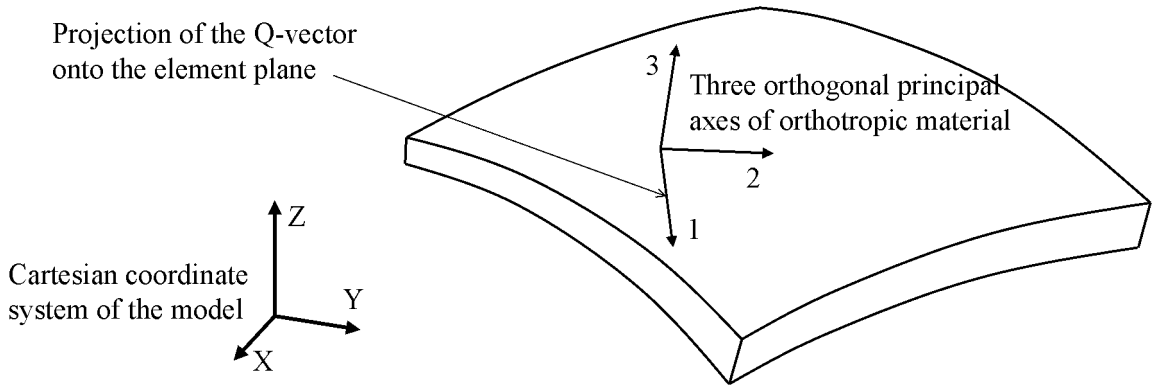


Figure 5.44 Shell element with orthotropic material properties

The elasticity matrix of an orthotropic material expressed in terms of the Young's and shear moduli and Poisson's ratio is given below.

The stress-strain relationship $\sigma = \mathbf{D} \cdot \varepsilon$ is:

$$\begin{bmatrix}\sigma_1\\ \sigma_2\\ \tau_{12}\\ \tau_{23}\\ \tau_{31}\end{bmatrix} = \frac{1}{1-\nu_{12}\nu_{21}}\begin{bmatrix} E_1 & \nu_{21}E_1 & 0 & 0 & 0\\ \nu_{12}E_2 & E_2 & 0 & 0 & 0\\ 0 & 0 & G_{12}(1-\nu_{12}\nu_{21}) & 0 & 0\\ 0 & 0 & 0 & G_{23}(1-\nu_{12}\nu_{21}) & 0\\ 0 & 0 & 0 & 0 & G_{31}(1-\nu_{12}\nu_{21})\end{bmatrix}\begin{bmatrix}\varepsilon_1\\ \varepsilon_2\\ \gamma_{12}\\ \gamma_{23}\\ \gamma_{31}\end{bmatrix} \quad (5.6)$$

where:

- ε_i, σ_i and E_i are the strain, stress and Young's modulus, respectively, in the direction of axis no. i.
- ν_{ij} is the coefficient expressing the negative strain in the direction of axis no. j caused by a positive strain in the direction of axis no. i ($\varepsilon_j = -\nu_{ij}\varepsilon_i$ for $\sigma_i = \sigma$ and all other stresses equal to zero).
- γ_{ij} (= γ_{ji}), τ_{ij} (= τ_{ji}) and G_{ij} (= G_{ji}) are the shear strain, shear stress and shear modulus, respectively, in the plane defined by axes nos. i and j.

Hence, the terms of the elasticity matrix becomes:

d11 = $E_1 / (1-\nu_{12}\nu_{21})$

d21 = $\nu_{21} E_1 / (1-\nu_{12}\nu_{21}) = \nu_{12} E_2 / (1-\nu_{12}\nu_{21})$

d22 = $E_2 / (1-\nu_{12}\nu_{21})$

d31 = d32 = 0

d33 = G_{12}

d41 = d42 = d43 = 0

d44 = G_{23}

d51 = d52 = d53 = d54 = 0

d55 = G_{31}

The material constants Young's modulus (E) and Poisson's ratio (ν) in the three directions are related as follows:

$\nu_{21} E_1 = \nu_{12} E_2$

$\nu_{31} E_1 = \nu_{13} E_3$

$\nu_{32} E_2 = \nu_{23} E_3$

where the second and third relations are irrelevant for the 3D-shell orthotropic material.

PROPERTY MATERIAL material-name ANISOTROPIC SOLID-ELEMENT

...	SOLID-ELEMENT	GLOBAL		rho ...
		TRANSFORMED	trnam	

...	d11	d21	d22	d31	d32	d33	d41	d42	d43	d44	d51	d52	d53	d54	d55	...

...	d61	d62	d63	d64	d65	d66	damp1	damp2	damp3	alpha1	alpha2	alpha3

PURPOSE:

The command specifies the data for either an anisotropic or orthotropic material for solid elements.

The anisotropic material has three material axes which in general will not constitute an orthogonal system (if so it will be an orthotropic material).

PARAMETERS:

GLOBAL	The axes of the anisotropic elasticity matrix are parallel with the cartesian coordinate system of the model.
TRANSFORMED	The material axes of the anisotropic elasticity matrix are parallel with the specified transformed coordinate system.
trnam	Name of a previously defined transformation.
rho	Material density.
d11 ... d66	Terms of the lower triangular part of the general anisotropic elasticity matrix; see Equation (5.7).
damp1 damp2 damp3	Specific damping coefficients for the three material axes of anisotropy.
alpha1 alpha2 alpha3	Thermal expansion coefficients for the three material axes of anisotropy.

$$\begin{bmatrix} d11 & & & & & \\ d21 & d22 & & & & \\ d31 & d32 & d33 & & & \\ d41 & d42 & d43 & d44 & & \\ d51 & d52 & d53 & d54 & d55 & \\ d61 & d62 & d63 & d64 & d65 & d66 \end{bmatrix}$$

The lower triangular general anisotropic elasticity matrix (5.7)

NOTES:

An orthotropic material is defined by taking advantage of the fact that it is a special case of the anisotropic material. To define an orthotropic material in its principal axes the lower triangular part of the general anisotropic elasticity matrix should have zero values for some of the terms:

$$\begin{bmatrix} d11 & & & & & \\ d21 & d22 & & & & \\ d31 & d32 & d33 & & & \\ 0 & 0 & 0 & d44 & & \\ 0 & 0 & 0 & 0 & d55 & \\ 0 & 0 & 0 & 0 & 0 & d66 \end{bmatrix} \tag{5.8}$$

PROPERTY MATERIAL material-name DAMPER

...	DAMPER	AXIAL	damp			
		TO-GROUND	n	{	cij	}*p

PURPOSE:

The command defines the damping coefficient of an axial damper and the damping matrix of a damper-to-ground.

PARAMETERS:

AXIAL — Define an axial damper.

damp — Axial damping coefficient.

TO-GROUND — Define a damper-to-ground.

n — Number of degrees of freedom of the damping matrix.

cij — Terms of the lower triangular part of the damping matrix. The terms are given column by column, i.e. c11 c21 ... cn1 c22 ... cn2 ... cnn; see the matrix below. The number of terms to enter, p, is a function of the number of degrees of freedom of the damping matrix, n, as follows:

$p = (n^2 + n)/2$.

The default value of the terms outside the diagonal is zero.

$$\begin{bmatrix} c11 & & & \\ c21 & c22 & & \\ \vdots & \vdots & & \\ cn1 & cn2 & \dots & cnn \end{bmatrix}$$

PROPERTY MATERIAL material-name ELASTIC

...	ELASTIC	young	poiss	rho	damp	alpha

PURPOSE:

The command defines the data for an isotropic elastic material.

PARAMETERS:

young	Young's modulus
poiss	Poisson's ratio
rho	Density
damp	Specific damping
alpha	Thermal expansion coefficient

PROPERTY MATERIAL material-name MASS

...	MASS	ONE-NODED	n	{	mij	}*p

PURPOSE:

The command defines the mass matrix of a mass element.

PARAMETERS:

ONE-NODED — Define a mass matrix for one node mass element.

n — Number of degrees of freedom of the mass matrix.

mij — Terms of the lower triangular part of the mass matrix. The terms are given column by column, i.e. m11 m21 ... mn1 m22 ... mn2 ... mnn; see the matrix below. The number of terms to enter, p, is a function of the number of degrees of freedom of the damping matrix, n, as follows:

$p = (n^2 + n)/2.$

The default value of the terms outside the diagonal is zero.

$$\begin{bmatrix} m11 & & & \\ m21 & m22 & & \\ \vdots & \vdots & & \\ mn1 & mn2 & \dots & mnn \end{bmatrix}$$

PROPERTY MATERIAL material-name SPRING

...	SPRING	AXIAL	spring				
		TO-GROUND	n	STIFFNESS / FLEXIBILITY	{	kij	}*p

PURPOSE:

The command defines the spring constant of an axial spring and the stiffness or flexibility matrix of a spring-to-ground. The flexibility matrix is the inverse of the stiffness matrix.

PARAMETERS:

AXIAL — Define an axial spring.

spring — Axial spring constant.

TO-GROUND — Define a spring-to-ground.

n — Number of degrees of freedom of the stiffness/flexibility matrix.

STIFFNESS — Stiffness matrix is to be given.

FLEXIBILITY — Flexibility matrix is to be given.

kij — Terms of the lower triangular part of the stiffness/flexibility matrix. The terms are given column by column, i.e. k11 k21 ... kn1 k22 ... kn2 ... knn; see the matrix below. The number of terms to enter, p, is a function of the number of degrees of freedom of the stiffness matrix, n, as follows:

$p = (n^2 + n)/2$.

The default value of the terms outside the diagonal is zero.

$$\begin{bmatrix} k11 & & & \\ k21 & k22 & & \\ \vdots & \vdots & & \\ kn1 & kn2 & \dots & knn \end{bmatrix}$$

PROPERTY POINT-MASS

...	POINT-MASS	select-points	pmtx	pmty	pmtz	pmrx	pmry	pmrz

PURPOSE:

The command defines point masses in nodes. The point masses may only be applied in geometry points.

A point mass is in effect a diagonal mass matrix whereas the mass element (see the PROPERTY MATERIAL material-name MASS command) allows definition of off-diagonal terms.

The contribution to the mass matrix are allowed to differ between the degrees of freedom. However, giving the same mass for the three translational degrees of freedom will normally be the only physically meaningful choice. The ‘rotational mass’ for the three rotational degrees of freedom may often be set to zero as their influence on the analysis results will normally be insignificant.

PARAMETERS:

select-points	Select points. See Section 5.1 on how to perform a selection.
pmtx	Translational mass in the X-direction.
pmty	Translational mass in the Y-direction.
pmtz	Translational mass in the Z-direction.
pmrx	Rotational mass about the X-axis.
pmry	Rotational mass about the Y-axis.
pmrz	Rotational mass about the Z-axis.

NOTES:

Point masses are accumulated if repeatedly given for the same point.

PROPERTY SECTION

...	SECTION	section-name	BAR	...
			BOX	
			CHANNEL	
			DOUBLE-BOTTOM	
			GENERAL	
			I	
			L	
			PIPE	
			UNSYMMETRICAL-I	

PURPOSE:

The command defines cross sections for beam and truss elements. The program offers different cross section types: pipe, I, L, etc. In addition a general type section can be defined by entering the cross sectional area, moments of inertia, sectional moduli, etc.

The local x-axis of beam and truss elements always follows the neutral axis of the element. In Figure 5.45 through Figure 5.53 the x-axis is directed into the paper plane. The local y- and z-axes are determined by the PROPERTY LOCAL-COORDINATE-BEAM command.

PARAMETERS:

section-name	Cross section name. Assigned to the appropriate beam and truss elements by the CONNECT SECTION command.
BAR	Define a bar cross section.
BOX	Define a box cross section.
CHANNEL	Define a channel cross section.
DOUBLE-BOTTOM	Define a double-bottom cross section.
GENERAL	Define a general cross section.
I	Define an I (or H) cross section.
L	Define an L cross section.
PIPE	Define a pipe cross section.
UNSYMMETRICAL-I	Define an un-symmetrical I cross section.

NOTES:

The effect of curvatures of inner corners of the cross sections are not taken into account in the calculation of the sectional properties (torsional moment of inertia).

Note that the dimensions of a cross section should not be such that the chosen cross section degenerates into another type. The formulae used for calculating the sectional properties are based on the assumption that the cross sections have reasonable shapes. Generally, the sectional moduli most sensitive to oddly shaped sections are the following:

- The torsional moment of inertia
- The shear centre location z-component of the UNSYMMETRICAL-I and L
- The shear area in the direction of the y-axis (SHARY) of the I and UNSYMMETRICAL-I

See Appendix B THEORY, Section B 1, for details on the formulae used for calculation of the sectional moduli.

PROPERTY SECTION section-name BAR

...	BAR	hz	bb	bt	sfy	sfz

PURPOSE:

The command defines a bar cross section.

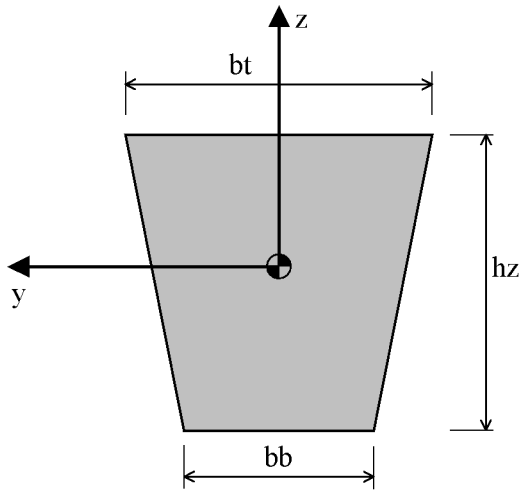


Figure 5.45 Bar section

PARAMETERS:

hz	Height
bb	Width at bottom
bt	Width at top
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters):

$SHARY_{modified} = SHARY_{program} \cdot sfy$

$SHARZ_{modified} = SHARZ_{program} \cdot sfz$

PROPERTY SECTION section-name BOX

...	BOX	hz	by	tt	ty	tb	sfy	sfz

PURPOSE:

The command defines a box cross section.

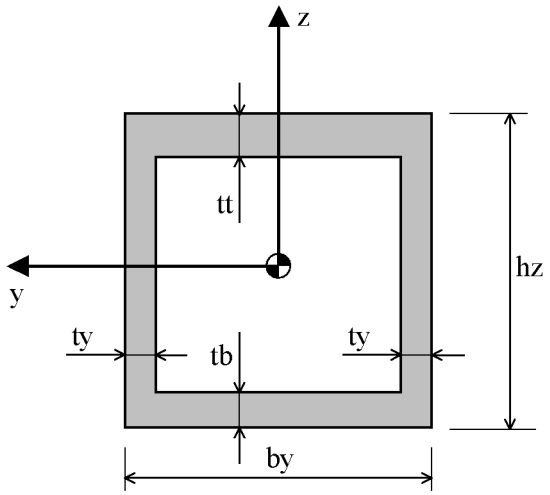


Figure 5.46 Box section

PARAMETERS:

hz	Height
by	Width
tt	Thickness of top flange
ty	Thickness of webs (vertical walls)
tb	Thickness of bottom flange
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters):

$SHARY_{modified} = SHARY_{program} \cdot sfy$

$SHARZ_{modified} = SHARZ_{program} \cdot sfz$

PROPERTY SECTION section-name CHANNEL

...	CHANNEL	hz	by	tz	ty	sfy	sfz	POSITIVE NEGATIVE

PURPOSE:

The command defines a channel cross section.

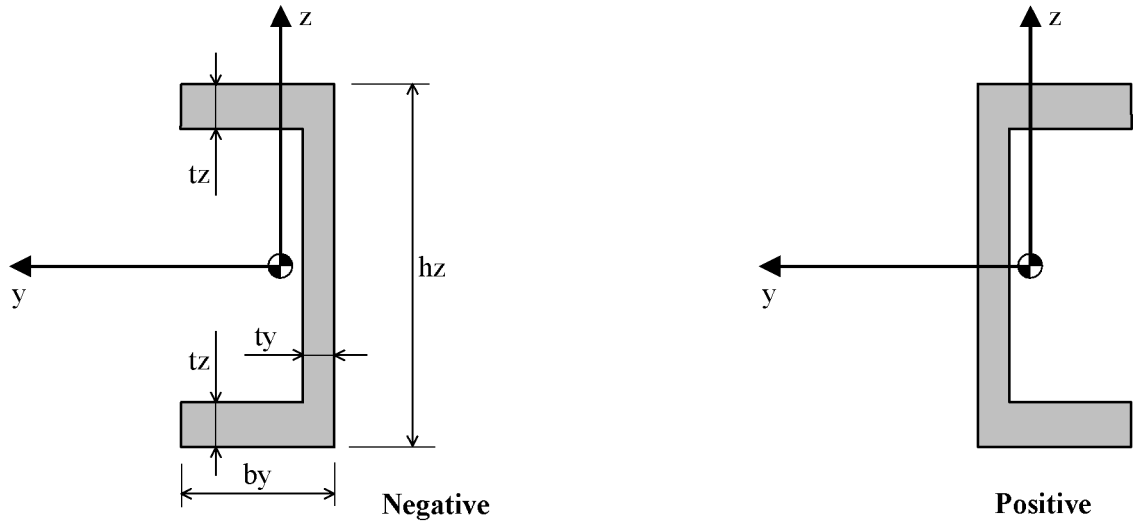


Figure 5.47 Channel section

PARAMETERS:

hz	Height
by	Width of top and bottom flanges
tz	Thickness of top and bottom flanges
ty	Thickness of web
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters): $SHARY_{modified} = SHARY_{program} \cdot sfy$ $SHARZ_{modified} = SHARZ_{program} \cdot sfz$
POSITIVE NEGATIVE	Web location in the local y-direction

PROPERTY SECTION section-name DOUBLE-BOTTOM

...	DOUBLE-BOTTOM	hz	ty	tb	tt	by	sfy	sfz

PURPOSE:

The command defines a double-bottom type cross section.

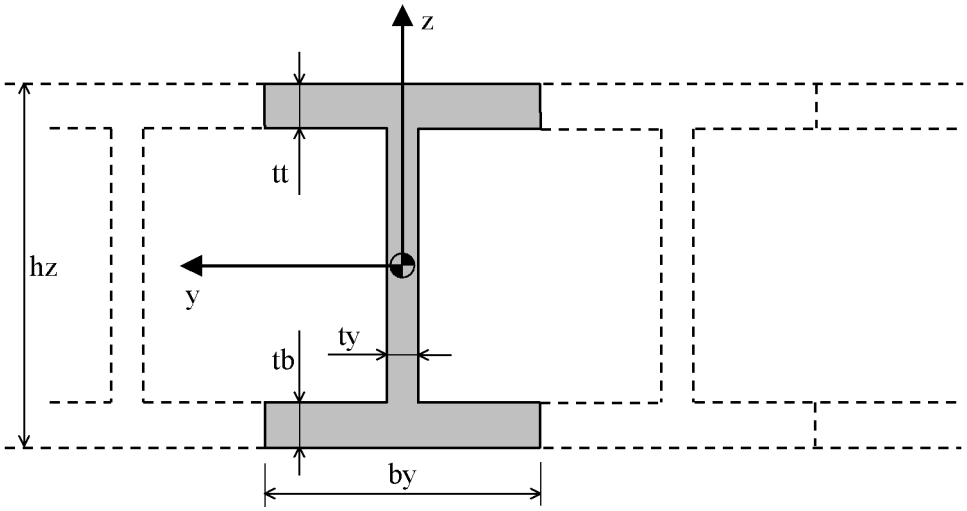


Figure 5.48 Double-bottom section

PARAMETERS:

hz	Height
ty	Thickness of web
tb	Thickness of bottom flange (plate)
tt	Thickness of top flange (plate)
by	Effective width of plates
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters):

$SHARY_{modified} = SHARY_{program} \cdot sfy$

$SHARZ_{modified} = SHARZ_{program} \cdot sfz$

NOTES:

The double-bottom section should be used with care. This is because the sectional properties are calculated in the same way as for the symmetrical I section. Only the torsional moment of inertia is increased to take into account the shear flow along the top and bottom plates. The effects in other directions are not considered.

PROPERTY SECTION section-name GENERAL

...	GENERAL	area	ix	iy	iz	iyz	wxmin	wymin	wzmin	...

...	shary	sharz	shceny	shcenz	sy	sz

PURPOSE:

The command defines a general section. All sectional data are defined directly. The following should be noted:

- For beams the area and moments of inertia are required while only the area is required for trusses.
- The product of inertia (I_{YZ}) is zero for all bi-symmetrical sections.
- The minimum sectional moduli (W_{Xmin}, W_{Ymin} and W_{Zmin}) are required by FRAMEWORK.
- Shear deformations will not be accounted for if the shear areas are zero.
- The shear centre location must be specified if the shear centre does not coincide with the element axis.
- The static area moments are used in connection with un-symmetrical sections ($I_{YZ}\neq 0$) to re-compute section values to the neutral axis.

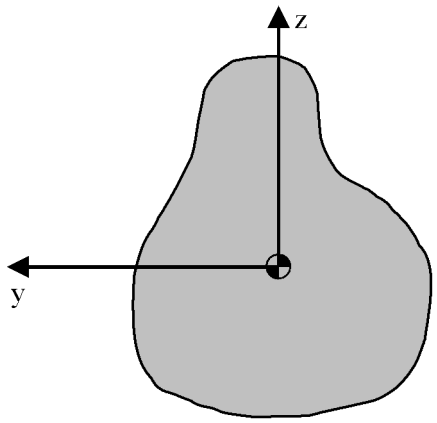


Figure 5.49 General cross section

PARAMETERS:

area	Cross sectional area (> 0)
ix	Torsional moment of inertia about shear centre (> 0)
iy	Moment of inertia about y-axis (> 0)

iz	Moment of inertia about z-axis (> 0)
iyz	Product of inertia about y- and z- axes
wxmin	Minimum torsional sectional modulus about shear centre (≥ 0)
wymin	Minimum sectional modulus about y-axis (≥ 0)
wzmin	Minimum sectional modulus about z-axis (≥ 0)
shary	Shear area in the direction of the y-axis (≥ 0)
sharz	Shear area in the direction of the z-axis (≥ 0)
shceny	Shear centre location from centroid, the y-component
shcenz	Shear centre location from centroid, the z-component
sy	Static area moment about y-axis (≥ 0)
sz	Static area moment about z-axis (≥ 0)

PROPERTY SECTION section-name I

...	I	hz	bt	tt	ty	bb	tb	sfy	sfz

PURPOSE:

The command defines a symmetrical I (or H) cross section.

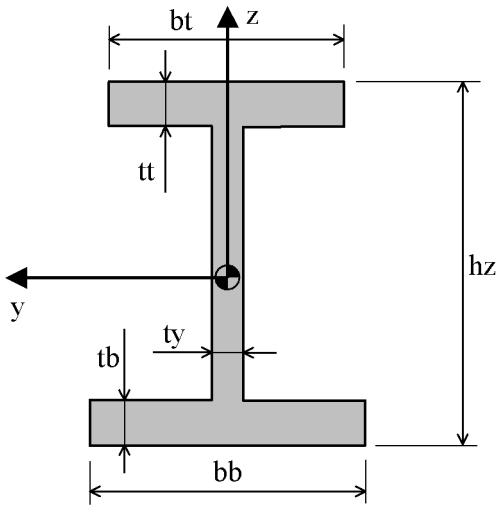


Figure 5.50 I section

PARAMETERS:

hz	Height
bt	Width of top flange
tt	Thickness of top flange
ty	Thickness of web
bb	Width of bottom flange
tb	Thickness of bottom flange
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters):

$SHARY_{modified} = SHARY_{program} \cdot sfy$

$SHARZ_{modified} = SHARZ_{program} \cdot sfz$

PROPERTY SECTION section-name L

...	L	hz	ty	by	tz	sfy	sfz	POSITIVE
								NEGATIVE

PURPOSE:

The command defines an L cross section.

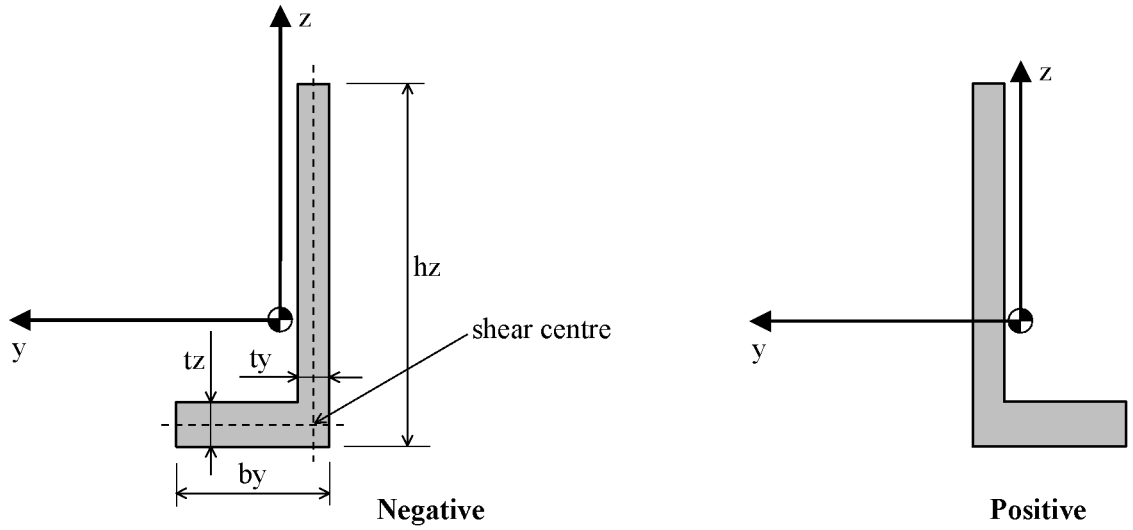


Figure 5.51 L section

PARAMETERS:

hz	Height
ty	Thickness of web
by	Width of flange
tz	Thickness of flange
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters): $SHARY_{modified} = SHARY_{program} \cdot sfy$ $SHARZ_{modified} = SHARZ_{program} \cdot sfz$
POSITIVE NEGATIVE	Web location in the local y-direction

PROPERTY SECTION section-name PIPE

...	PIPE	dy	t	sfy	sfz

PURPOSE:

The command defines a pipe cross section.

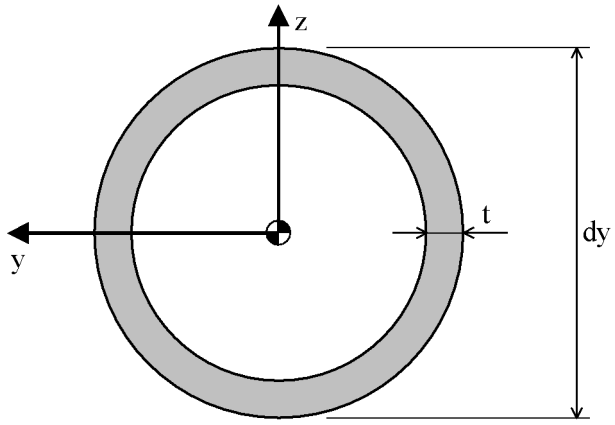


Figure 5.52 Pipe section

PARAMETERS:

dy — Outer diameter

t — Thickness of wall

sfy sfz — Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters):

$SHARY_{modified} = SHARY_{program} \cdot sfy$

$SHARZ_{modified} = SHARZ_{program} \cdot sfz$

PROPERTY SECTION section-name UNSYMMETRICAL-I

...	UNSYMMETRICAL-I	hz	bt	b1	tt	ty	bb	b2	tb	sfy	sfz

PURPOSE:

The command defines an un-symmetrical I cross section.

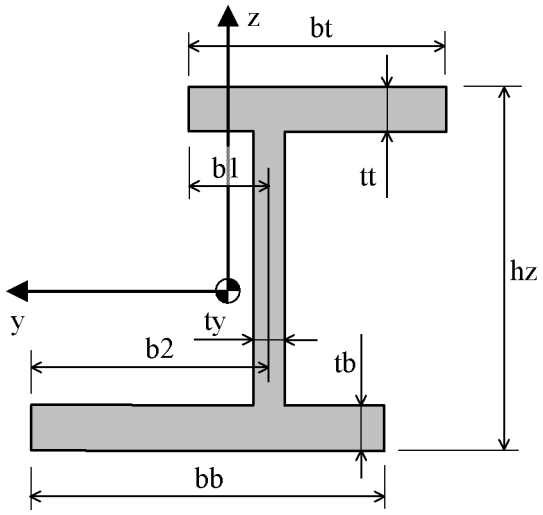


Figure 5.53 Un-symmetrical I section

PARAMETERS:

hz	Height
bt	Width of top flange
b1	Width of part of top flange along positive local y-axis
tt	Thickness of top flange
ty	Thickness of web
bb	Width of bottom flange
b2	Width of part of bottom flange along positive local y-axis
tb	Thickness of bottom flange
sfy sfz	Factors modifying the shear areas calculated by the program. The modified shear areas are (see the PRINT SECTION command for an explanation of the parameters):

$$SHARY_{modified} = SHARY_{program} \cdot sfy$$

$$SHARZ_{modified} = SHARZ_{program} \cdot sfz$$

PROPERTY THICKNESS

...	THICKNESS	select-surfaces	thickness

PURPOSE:

The command specifies element thicknesses (membrane and shell elements) by assigning thicknesses to geometry surfaces.

PARAMETERS:

select-surfaces Select surfaces. See Section 5.1 on how to perform a selection.

thickness Thickness of the selected surfaces.

NOTES:

Rather than giving a constant value for the thickness a function describing a thickness variation may be given; see Section 5.3.

The function ONLY-BETWEEN is less suitable for specifying thicknesses and should be avoided.

PROPERTY TRANSFORMATION

...	TRANSFORMATION	trnam	ARBITRARY	point1	point2	angle					
			GUIDING-POINT	spx	spy	spz		gpx	gpy	gpz	
			ROTATION	GLOBAL		...	X	angle			
				LOCAL			Y				
							Z				

PURPOSE:

The command defines a geometrical transformation consisting of pure rotations of the cartesian coordinate system. Transformations are used in copying geometric entities, for defining boundary conditions and loads in transformed (askew) coordinate systems and for orientating spring-to-ground and damper-to-ground elements.

Alternatively to this command the DEFINE TRANSFORMATION command may be used. The user may find that solely using the DEFINE TRANSFORMATION command is best as it is more powerful than the PROPERTY TRANSFORMATION command.

PARAMETERS:

trnam — User-given name of the transformation matrix.

ARBITRARY — Define a rotation about an axis defined by two points.

point1 point2 — Two points defining the axis. Positive rotation is according to the right-hand-rule.

angle — Rotation angle. Positive direction is defined by the right-hand-rule.

GUIDING-POINT — Use guiding points to define a rotation. A second-point (SP) and a guiding-point (GP) are defined. The x-axis of the transformed coordinate system (X_T) goes from the origin and through point SP. Point GP lies in the X_T-Z_T-plane on the positive Z_T side. Y_T forms together with X_T and Z_T a right-handed cartesian coordinate system. See Figure 5.54.

spx spy spz — Global coordinates of the second-point.

gpx gpy gpz — Global coordinates of the guiding-point.

ROTATION — Define a rotation about given axes. One or more rotations about the GLOBAL or LOCAL axes may be defined. Rotations about the GLOBAL axes refer to the fixed cartesian coordinate system of the model. Rotations about the LOCAL axes refer to the axes of the current transformed coordinate system.

GLOBAL — A rotation is performed about an axis of the cartesian coordinate system of the model.

LOCAL — A rotation is performed about an axis of the current transformed coordinate system.

X Y Z The rotation is about the X, Y or Z axis, respectively.

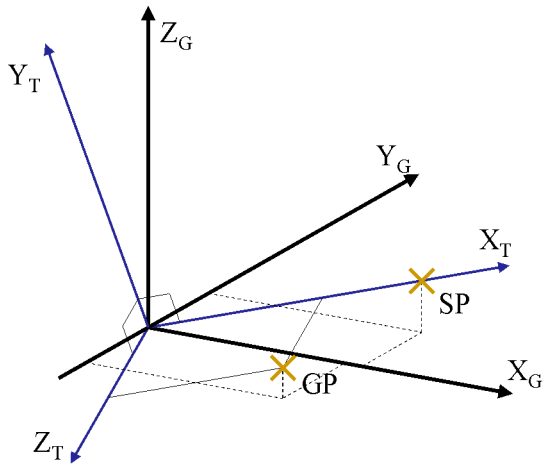


Figure 5.54 Defining a transformation using the GUIDING-POINT option

RE-COMPUTE

RE-COMPUTE	LOADS

PURPOSE:

The command re-computes or re-distributes the loads to nodes and elements.

When changes have been made to the loads or mesh this command may be used to re-distribute the loads. A load display may otherwise be incorrect.

A re-computation will always be performed when the Input Interface File is produced.

PARAMETERS:

LOADS Redistributes the loads

RE-DISPLAY

RE-DISPLAY	MESH
	GEOMETRY
	ALL

PURPOSE:

The command re-displays the current view (refreshes the screen). This is relevant for example after changing the viewing position (using the commands SET GRAPHICS EYE-DIRECTION or ROTATE) and to remove the previous load display after adding a new one.

PARAMETERS:

MESH	Re-display only the mesh.
GEOMETRY	Re-display only the geometry.
ALL	Re-display both mesh and geometry.

READ

READ	DXF	file-prefix	file-name	
	DXF-SETTINGS	INCLUDE-THICKNESS		NO
				YES
		MAX-MESH-CORNER-ANGLE		angle
		POLYLINE-MODE	CLOSED-TO-SURFACE	NO
				YES
			TO-LINES	
			TO-NODELINE	
			TO-SPLINE	
	INPUT-INTERFACE-FILE	file-prefix	superelement-number	
	JOURNALLING-MODE	ON		
		OFF		

PURPOSE:

The command sets parameters for and reads data from file. There are two formats that can be read:

- SESAM Input Interface File (T#.FEM) containing a first level superelement

  The Input Interface File (T-file) contains a FE model (elements and nodes with properties) and no geometry information. Therefore, reading this file involves establishing a FE model in Prefem's database with no corresponding geometry. Currently, no modifications of the FE model may be done. The model can thus be used for verification purposes, only.

  The Input Interface File may have been generated by preprocessors other than Prefem. Prefem will still be able to print and display the data.

  When the reading has been completed a message giving the number of nodes, elements and load cases is printed, e.g.:

  83 NODES READ

  167 ELEMENTS READ

  5 LOADCASES READ

  Click the 'Display Mesh' and 'Zoom Fr' buttons to see the mesh.

- DXF-format (*.DXF) which is a file format for exchanging 3D CAD data (DXF originates from the CAD program AutoCAD)

  A DXF-file contains a number of items of which the following are interpreted and converted into Prefem data:

  — POINT

  — LINE

— VERTEX
— 2D-POLYLINE (sub-type of POLYLINE)
— 3D-POLYLINE (sub-type of POLYLINE)
— 3D-POLYGON-MESH (sub-type of POLYLINE, cubic B-spline surface type)
— ARC
— 3DLINE
— 3DFACE
— INSERT
— BLOCK section (referred to by INSERT)

The BLOCK section typically consists of other DXF entities. The INSERT entity contains an insert position and scaling factor. This enables deployment of the same BLOCK at different positions and possibly with different scaling factors.

PARAMETERS:

DXF	Read a geometry model stored in a DXF-formatted file and create a Prefem geometry model. The geometric entities point, line and surface are imported. A log-file is generated during the reading. The log has file type '.LOG' and will have the same name as the DXF-file.
file-prefix	File name prefix.
file-name	File name excluding the mandatory file extension (.DXF for a DXF-file and .FEM for a SESAM Input Interface File).
DXF-SETTINGS	Set control parameters for reading the DXF-file. Note that these settings are lost when a READ command has been executed.
INCLUDE-THICKNESS	Control whether the thickness information on the DXF-file is to be read (YES) or not (NO). Note that on the DXF-file the thickness information is unique for each surface. Default setting is NO.
MAX-MESH-CORNER-ANGLE	Set the maximum angle for a surface corner to be defined as mesh-corner. Corners having a larger angle will be defined as not-mesh-corner. The default value is 150 degrees.
angle	Angle in degrees
POLYLINE-MODE	Control how the DXF polyline is interpreted and also whether to create Prefem surfaces from closed polylines. These settings are reset after the READ command.
CLOSED-TO-SURFACE	Control whether all closed 2D and 3D polylines shall be converted to surfaces (YES) or not (NO). All line segments of a polyline will be converted to Prefem lines. Default setting is NO.

TO-LINES	All line segments of a polyline will be converted to Prefem lines. This is the default choice.
TO-NODELINE	Polylines will be converted to Prefem node-lines. This is the default choice (among TO-LINES, TO-NODELINE and TO-SPLINE).
TO-SPLINE	Polylines will be converted to Prefem splines. Note that this conversion is not necessarily shape preserving, i.e. the spline curve may differ from the corresponding polyline.
INPUT-INTERFACE-FILE	Read a FE model stored in a SESAM Input Interface File.
superelement-number	Number of the superelement to be read. The file with the following name will be read: file-prefixTsuperelement-number.FEM
JOURNALLING-MODE	Switch on/off a mode involving logging of Prefem commands for creating the geometry and property (thickness). This involves a translation of the information on the DXF-file into Prefem commands. These Prefem commands may be used to recreate the model at a later stage. By default this mode is off involving that the READ DXF command is logged. Thus, the command log (journal) file cannot later be used without access to the DXF-file. Note that the mode is set to off after the READ command.

ROTATE

ROTATE	X-AXIS	degrees
	Y-AXIS	
	Z-AXIS	

PURPOSE:

The command rotates the display. The rotations are about the axes specified by the command SET GRAPHICS ROTATION-MODE. The global axes of the model are the default axes.

You may find that rotating the model interactively is more efficient; see Section 3.1.

PARAMETERS:

degrees Angle in degrees.

SET

SET	COMMAND-INPUT-FILE	...
	DEFAULT	
	ELEMENT-LENGTH-RATIO	
	ELEMENT-TYPE	
	GRAPHICS	
	INSIDE	
	JOURNALLING	
	MAX-ELEMENT-LENGTH	
	MESH	
	MESH-CORNER	
	MESH-PARAMETERS	
	NAMING	
	NOT-MESH-CORNER	
	NUMBEROF-ELEMENTS	
	OUTSIDE	
	PLANE-STRAIN	
	PLANE-STRESS	
	PLOT	
	PREFIX-NAME	
	PRINT	
	PROJECTION	
	TASK	
	TOLERANCE	
	WRITE-MODE	

PURPOSE:

The command defines and sets various parameters.

PARAMETERS:

COMMAND-INPUT-FILE — Set name of the command input file. Use the # command to read commands. You may, however, find it more convenient to

	specify a command input file when starting Prefem from Manager.
DEFAULT	Set various default conditions, e.g. which property (type of element, number of elements, material, section, thickness, etc.) to automatically assign to subsequently created geometry.
ELEMENT-LENGTH-RATIO	Set length of elements (element edges), see Section 3.4.1.
ELEMENT-TYPE	Set types of element to be created for geometry (lines, surfaces and bodies); see Section 2.5 for information on element types.
GRAPHICS	Set various parameters controlling the graphic presentation.
INSIDE	Define inside of surfaces for the benefit of pressure load application, see Section 3.12.3.
JOURNALLING	Switch on logging in command log file (see Chapter 4) of print and graphics commands.
MAX-ELEMENT-LENGTH	Set maximum element length for lines, see Section 3.4.1.
MESH	Set control information for mesh, especially mesh-corner types for surfaces.
MESH-CORNER	Set mesh-corner for surfaces, the SET MESH command may also be used for the same purpose.
MESH-PARAMETERS	Set parameters controlling the creation of the finite element mesh.
NAMING	Control the new geometry names created when cutting geometry.
NOT-MESH-CORNER	Set not-mesh-corner for surfaces, the SET MESH command may also be used for the same purpose.
NUMBEROF-ELEMENTS	Set number of elements for lines, see Section 3.4.1.
OUTSIDE	Define outside of surfaces for the benefit of pressure load application, see Section 3.12.3.
PLANE-STRAIN	This feature is presently not active.
PLANE-STRESS	This feature is presently not active.
PLOT	Set name of plot file, plot format and other plot parameters.
PREFIX-NAME	Set a prefix for geometry names, a string that is put in front of subsequently created geometry names.
PRINT	Set print destination (screen or file?), name of print file and other parameters.

PROJECTION	Set projection onto a shape of mesh for selected surfaces.
TASK	Suppress commands irrelevant for the current task.
TOLERANCE	Set tolerances used by the program to determine whether two values are equal, e.g. in checking geometric coincidence.
WRITE-MODE	Decide whether or not defined sets are to stored on the Input Interface File.

SET COMMAND-INPUT-FILE

...	COMMAND-INPUT-FILE	file-prefix	file-name

PURPOSE:

The command specifies the name of the command input file. The file extension of the command input file must be .JNL.

The # command will read commands stored on the command input file into the program. The commands read will have the same effect as if they were entered from the keyboard. Several command input files may be used in sequence but only one command input file can be used at a time.

The command input file name cannot be the same as the command log (journal) file name. See Section 4.1.5 for information on the command input file.

You may, however, find it more convenient to specify a command input file when starting Prefem from Manager.

PARAMETERS:

file-prefix File name prefix.

file-name File name excluding the mandatory file extension (.JNL).

SET DEFAULT

...	DEFAULT	ADJUST-MESH		...	ON	
		AUTOMATIC-NAMING			OFF	
		COPY-ELEMENT-TYPE				
		ELEMENT-TYPE	LINE	...	element-type	
			SURFACE		NONE	
			BODY			
		EYE-DIRECTION			eyex	eyey eyez
		LINEAR-DEPENDENCY-MODE			FORCE-TO-SUPER	
					NO-FORCE-TO-SUPER	
		LOCAL-COORDINATE-BEAM			NONE	
					YX-PLANE	...
					ZX-PLANE	
		MATERIAL			material-name	
		MAX-ELEMENT-LENGTH			length	
		NUMBEROF-ELEMENTS			nelm	
		SECTION			section-name	
		THICKNESS			thickness	

PURPOSE:

The command sets various default conditions.

PARAMETERS:

ADJUST-MESH

Switch on/off automatic mesh adjustment in connection with changing the number of elements (or maximum element length) for lines. This involves automatic execution of the MESH ADJUST command. Note that the there is a Shortcut command button which toggles this automatic mesh adjustment.

AUTOMATIC-NAMING

Switch on/off automatic naming mode. This involves that rather than prompting the user Prefem will automatically generate geometry names when geometry is defined. The logging of the commands for defining geometry will include the automatically generated names.

COPY-ELEMENT-TYPE	Switch on/off whether type of element is going to be carried over when geometry is copied. See the COPY command.
ELEMENT-TYPE	Set type of element that will be assigned to subsequently created geometry.
LINE	Set default type of element for subsequently created lines.
SURFACE	Set default type of element for subsequently created surfaces.
BODY	Set default type of element for subsequently created bodies.
element-type	Choose desired type of element. See Section 2.5.
NONE	Any element type previously chosen for this geometry is annulled.
EYE-DIRECTION	Specify the default direction from origin to 'eye', i.e. the default viewpoint. See the SET GRAPHICS EYE-DIRECTION command for further explanation. This setting will overrule the setting when starting Prefem (typically given in Manager). After a rotation of the model the viewpoint will return to this default viewpoint when the command SET GRAPHICS EYE-DIRECTION DEFAULT is given (or the Direct access button Default is clicked).
eyex eyey eyez	X-, Y- and Z-components of the direction to the 'eye'.
LINEAR-DEPENDENCY-MODE	Choose linear dependency mode, i.e. whether independent nodes must be super or not. See Section 3.5.8 for an explanation.
FORCE-TO-SUPER	Independent nodes that are not already super are forced to become super.
NO-FORCE-TO-SUPER	Independent nodes need not be super.
LOCAL-COORDINATE-BEAM	A default local coordinate system for beam elements is given. This default is valid for subsequently created lines. **Note: Even without use of this command default local coordinate systems will be assigned to beams. This default corresponds to the setting of the command SET DEFAULT LOCAL-COORDINATE-BEAM ZX-PLANE / ZX-PLANE.** **Note: Copied lines will inherit the setting of the original. I.e. the setting of this command will not apply.** **Note: The default set by this command is overruled by explicit assignment using the PROPERTY LOCAL-COORDINATE-BEAM command.**

NONE	Any previously given default setting for local coordinate system for beams is annulled.
YX-PLANE	The local yx-plane is used for determining the local coordinate system. The command alternatives that follow correspond to the PROPERTY LOCAL-COORDINATE-BEAM command, see this.
ZX-PLANE	The local zx-plane is used for determining the local coordinate system. The command alternatives that follow correspond to the PROPERTY LOCAL-COORDINATE-BEAM command, see this.
MATERIAL	Set material name that will be assigned to subsequently created geometry. **Note: The default set by this command is overruled by explicit assignment using the CONNECT MATERIAL command.**
material-name	A previously defined material name.
MAX-ELEMENT-LENGTH	Set maximum element length that will be assigned to subsequently created lines. **Note: The default set by this command is overruled by explicit assignment using either the SET NUMBEROF-ELEMENTS command or the SET MAX-ELEMENT-LENGTH command.**
length	Element length
NUMBEROF-ELEMENTS	Set number of elements that will be assigned to subsequently created lines. The default is 4. **Note: The default set by this command is overruled by explicit assignment using either the SET NUMBEROF-ELEMENTS command or the SET MAX-ELEMENT-LENGTH command.**
nelm	Number of elements
SECTION	Set section name that will be assigned to subsequently created lines. **Note: The default set by this command is overruled by explicit assignment using the CONNECT SECTION command.**
section-name	A previously defined section name.

THICKNESS	Set thickness that will be assigned to subsequently created surfaces. **Note: The default set by this command is overruled by explicit assignment using the PROPERTY THICKNESS command.**
thickness	Thickness

SET ELEMENT-LENGTH-RATIO

...	ELEMENT-LENGTH-RATIO	select-lines	...

...	EQUALLY-SPACED			
	ARITHMETRIC-SEQUENCE	[starting-point]	relative-length-first-elem	relative-length-last-elem
	GIVEN-RELATIVE	[starting-point]	relative-length-of-elem	*

PURPOSE:

The command defines the length of the elements (or element edges) for selected lines. The command is irrelevant for lines of type node-line.

The element (edge) lengths may be equal, linearly increasing/decreasing and the individual elements (element edges) may be given relative lengths. The lengths relate to a starting-point chosen by the user. If more than one line is selected then the starting-point will not be requested rather the first point according to the definition of the line is taken as the starting-point.

Mid-side nodes (for higher order elements) will always be in the midpoint of the element (edge). (Except for node-lines which also determine the position of mid-side nodes.)

PARAMETERS:

select-lines	Select lines. See Section 5.1 on how to perform a selection.
EQUALLY-SPACED	All elements (element edges) will be of equal length.
ARITHMETRIC-SEQUENCE	The element (edge) length will vary as an arithmetric sequence.
starting-point	Point defining the starting point of the line for distributing the element lengths. Only given when a single line has been selected.
relative-length-first-elem	Relative element length of the first element, i.e. the element closest to the starting-point.
relative-length-last-elem	Relative element length of the last element.
GIVEN-RELATIVE	The relative element (edge) lengths are given individually.
relative-length-of-elem	Relative element length given as a positive number for each element on the line(s) starting at the starting-point. The number of values to give must correspond to the number of elements for the lines in question.

NOTES:

If the GIVEN-RELATIVE option has been used and the number of elements for the lines in question is thereafter *increased* then the extra elements will have no relative length value. This involves that element lengths will be zero, i.e. the elements (element edges) will collapse. To remedy this situation a new SET ELEMENT-LENGTH-RATIO command must be given. To avoid such a situation use the EQUALLY-SPACED or ARITHMETRIC-SEQUENCE options rather than the GIVEN-RELATIVE option.

On the other hand, if the number of elements for lines is *decreased* after giving relative lengths of all elements for lines then the superfluous length values will be neglected.

SET ELEMENT-TYPE

...	ELEMENT-TYPE	LINE	select-geometry	...	element-type
		SURFACE			NONE
		BODY			

PURPOSE:

The command controls the types of elements created by the MESH (or CREATE MESH) command. The element types available are described in Section 2.5.

PARAMETERS:

LINE	Choose element type for lines (1-D elements).
SURFACE	Choose element type for surfaces (2-D elements).
BODY	Choose element type for bodies (3-D elements).
select-geometry	Select appropriate geometry. See Section 5.1 on how to perform a selection.
element-type	Choose desired type of element. See Section 2.5.
NONE	Any element type previously chosen for this geometry is annulled.

NOTES:

Note that spring, damper and mass elements are not set by this command. See the DEFINE command for these elements.

Most element types have six d.o.f.s but some have only three (truss and solid elements). It is possible to use elements with six d.o.f.s in combination with elements with only three d.o.f.s. The common nodes will have six d.o.f.s.

SET GRAPHICS

...	GRAPHICS	AUTOMATIC	ON		
			OFF		
		CHARACTER-TYPE	HARDWARE		
			SOFTWARE		
		COLOUR	...		
		CURVE-DRAW	FINE		
			NORMAL		
			COARSE		
		DEVICE	device-name		
		EYE-DIRECTION	eyex	eyey	eyez
			DEFAULT		
		HIDDEN	ON		
			OFF		
		INPUT	ON		
			OFF		
		NUMERICAL-VALUES	...		
		PLOT-FILE	file-prefix	file-name	
		PRESENTATION	...		
		PROPERTY-SELECTION-AUTO	ON		
			OFF		
		ROTATION-MODE	GLOBAL-AXES		
			SCREEN-AXES		
		SCALE	scale		
			AUTO		
		SHRINK-FACTOR	shrink-factor		
		SIZE-SYMBOLS	...		
		VIEWPORTS	number-of-viewports		

PURPOSE:

The command sets control parameters for the graphical features of the program.

PARAMETERS:

AUTOMATIC	Switch automatic displaying of all geometry created or changed ON or OFF. Default for graphical user interface is ON and default for line-mode user interface is OFF.
CHARACTER-TYPE	Choose between hardware and software type characters. Hardware characters are faster to draw, but software characters may be better shaped and their sizes may be adjusted by the SET GRAPHICS SIZE-SYMBOLS command. Software characters are default.
COLOUR	Set multiple colours for display of load and thickness. See separate explanation of the command.
CURVE-DRAW	Choose accuracy of drawing of curves. For small models and when zooming in on a large model the FINE alternative may be useful. For large models where displaying the model takes time the COARSE alternative will speed it up. NORMAL is the normal (default) accuracy.
DEVICE	Choose appropriate type of graphics device. Legal alternatives will depend on the hardware you are using. X-WINDOW is the default choice on most Unix computers while WINDOWS is the default choice for PCs.
EYE-DIRECTION	Set the viewpoint for the model display. The command specifies the direction (vector) from origin to 'eye', i.e. the viewpoint. The 'eye' will be positioned on the line through the origin and the point (eyex, eyey, eyez) at an infinite distance from the origin thus resulting in a parallel projection. By changing the eye direction the model is displayed from different angles. Giving DEFAULT rather than a point sets the viewpoint back to what it was when entering Prefem (the setting in Manager). You may find that rotating the model interactively is more efficient; see Section 3.1.
HIDDEN	Switch ON and OFF hidden display mode. Hidden mode requires more computation time. For large models combined with limited computer resources this may be a problem. The hidden mode is only relevant for display of FE mesh, i.e. it has no effect on display of geometry model. Note that Shift+Esc will abort a time consuming display; see Section 4.2.1.
INPUT	Switch between graphical (option ON) and line-mode (option OFF) user interface. See Chapter 4.

NUMERICAL-VALUES	Choose format and number of decimals for numerical values displayed. See separate explanation of the command.
PLOT-FILE	Set name of plot file. Any number of plots may be sent to the same plot file. If a new plot file name is given then the previous one is closed. Note that if you want to import a plot into a word processor then you may need to limit each plot file to a single plot. This command has the same effect as the SET PLOT FILE command.
PRESENTATION	Choose between various graphic presentation modes for elements, etc. See separate explanation of the command.
PROPERTY-SELECTION-AUTO	Set automatic display of geometry selected within a PROPERTY command. With this switch ON the geometry will be displayed with dotted lines when hitting carriage within a PROPERTY command at the point where geometry is to be selected. Then, selecting geometry and hitting carriage return will highlight the geometry selected by solid lines. The command has no effect for the graphical user interface in which case colour highlighting is used for the same purpose.
ROTATION-MODE	Choose between GLOBAL-AXES and SCREEN-AXES for the ROTATE command. The x-axis of the SCREEN-AXES is horizontal and pointing to the right while the y-axis is vertical and pointing up. By default GLOBAL-AXES are used.
SCALE	Determine manually by which scale the model is displayed. The AUTO option will calculate a scale that, independent of the viewing direction, best adapts the display to the available plotter/screen area. If a zoomed (in or out) view is shown the AUTO option will return to fitting the complete model into the screen area (the same is achieved by the ZOOM OFF command). To determine a user specified scale of reasonable magnitude notice that by entering the SET GRAPHICS SCALE command the program will give as default value for the scale the one calculated by the AUTO option.
SHRINK-FACTOR	Set a shrink factor for display of elements. The legal range is 0.1 - 1.0. Switching off shrunken mode is done by giving the value 1.0.
SIZE-SYMBOLS	Change the sizes of symbols like geometry and mesh names, etc. See separate explanation of the command.
VIEWPORTS	Choose between 1, 2 and 4 viewports.

Two viewports will initially display the model in both viewports. The right-hand viewport is then available for zooming. The left-hand viewport will display the whole model and a broken square indicating the zoomed-in area. (Zooming may also be performed in the left-hand viewport but the right-hand viewport will then not indicate the zoomed-in area by a broken square.)

If the model has already been zoomed when switching to two viewports then the right-hand viewport will show the zoomed view and the left-hand viewport the whole model with a broken square indicating the zoomed-in area.

Switching to four viewports will display the model projected into the X-Z-, Y-Z- and X-Y-planes. The fourth viewport will display the current model and viewpoint. Zooming is possible in all viewports.

SET GRAPHICS COLOUR

...	COLOUR	LOAD-VALUE-LEVELS	number-of-levels			
		BODY-NAMES	...	BLACK	...	LIGHT
		BOUNDARY-CONDITIONS				
		ELEMENT-NUMBERS		BLUE		
		GEOMETRY-LINES				
		LINE-NAMES		GRAY		
		LOAD-VALUES				MEDIUM
		LOCATE-LINES		GREEN		
		MATERIAL-NAME				
		MESH		ORANGE		
		MESH-CORNERS				
		NODE-NUMBERS		RED		
		NODE-SYMBOLS				DARK
		POINT-NAMES		VIOLET		
		POINTS				
		SUPER-NODE-SYMBOLS		WHITE		
		SURFACE-NAMES		YELLOW		

PURPOSE:

The command sets colour for lines, names, numbers and symbols.

The command also sets multiple colours for load arrows. The command SET GRAPHICS PRESENTATION LOAD ARROW (not CONNECTED-ARROWS) is required to use this feature.

PARAMETERS:

LOAD-VALUE-LEVELS	Set number of differently coloured load value levels.
number-of-levels	Number of different load value levels.
BODY-NAMES	Set colour for body names, default colour is white.
BOUNDARY-CONDITIONS	Set colour for boundary condition symbols, default colour is medium blue.
ELEMENT-NUMBERS	Set colour for element numbers, default colour is medium blue.

GEOMETRY-LINES	Set colour for geometry lines, default colour is medium green.
LINE-NAMES	Set colour for line names, default colour is medium red.
LOAD-VALUES	Set colour for load values, default colour is medium green.
LOCATE-LINES	Set colour for lines appearing when using the LOCATE command, default colour is medium green.
MATERIAL-NAME	Set colour for material names, default colour is the same as the relevant geometry name (line, surface, body).
MESH	Set colour for FE mesh (elements), default colour is medium red.
MESH-CORNERS	Set colour for mesh corner symbols, default colour is medium violet.
NODE-NUMBERS	Set colour for node numbers, default colour is medium green.
NODE-SYMBOLS	Set colour for free node symbols (the diamond for nodes with no boundary condition), default colour is medium yellow.
POINT-NAMES	Set colour for point names, default colour is light blue.
POINTS	Set colour for point symbols (the X), default colour is medium yellow.
SUPER-NODE-SYMBOLS	Set colour for supernodes (nodes for which one or more d.o.f.s are defined as super), default colour is medium blue.
SURFACE-NAMES	Set colour for surface names, default colour is medium violet.
BLACK WHITE	For these two ‘colours’ there is no choice between light, medium and dark.
BLUE ... YELLOW	For these colours there is a choice between light, medium and dark.
LIGHT MEDIUM DARK	Choose colour intensity. LIGHT may be the best choice for black background (screen) and DARK the best for white background (plots).

SET GRAPHICS NUMERICAL-VALUES

...	NUMERICAL-VALUES	FORMAT	E-FORMAT
			F-FORMAT
		NUMBER-OF-DECIMALS	number-of-decimals

PURPOSE:

The command chooses format and number of decimals for numerical values shown in the display.

By default, two decimals and F-format is used.

PARAMETERS:

FORMAT	Select format for numerical values.
E-FORMAT	E-format is selected. Example with two decimals: 0.12E+03
F-FORMAT	F-format is selected. Example with two decimals: 123.00
NUMBER-OF-DECIMALS	Set the number of decimals to use for numerical values. Example with three decimals and E-format: 0.123E+03.
number-of-decimals	The number of decimals.

SET GRAPHICS PRESENTATION

...	PRESENTATION	BEAM-ELEMENT	BETWEEN-NODES	
			DRAWN-ECCENTRIC	
			LINE-WIDTH	line-width-scaling
			OUTLINE-SECTION	
			SECTION-AS-SOLID	
			SHOW-ECCENTRICITY	ON
				OFF
			SIMPLE-SECTION	
		ELEMENT-THICKNESS	NUMERICAL	
			SYMBOLIC	
		FILLED-ELEMENTS	ON	
			OFF	
		LAYERED-ELEMENTS	LAYERS	
			LOCAL-AXIS	
			OUTLINE-AREA	
			OUTLINE-SECTION	
			SHELL-ONLY	
			SOLID-AREA	
			SOLID-SECTION	
		LOAD	ARROW	
			CONNECTED-ARROWS	
			NUMERICAL	
		MIDNODE-ELEMENTS	NORMAL	
			SIMPLIFIED	

PURPOSE:

The command chooses between various graphic presentation modes for labelling of element thicknesses, displaying beam elements, displaying loads, etc.

PARAMETERS:

BEAM-ELEMENT Select display mode for beam elements.

BETWEEN-NODES

Draw the beam elements as straight lines between the nodes.

DRAWN-ECCENTRIC

Draw the beam elements as straight lines taking their eccentricities into account. The display of eccentricities is enlarged by the SET GRAPHICS SIZE-SYMBOLS SECTION-FACTOR command.

LINE-WIDTH line-width-scaling

Beam elements are by default drawn with a line three times thicker than other lines (line-width-scaling = 3). With this option you may set the line thickness of beam elements equal to standard (line-width-scaling = 1) and increase it up to 25 times the standard thickness.

OUTLINE-SECTION

Draw the beam elements with their sections fully drawn and taking their eccentricities into account. The display of eccentricities is enlarged by the SET GRAPHICS SIZE-SYMBOLS SECTION-FACTOR command.

The general section for which only area and sectional properties are given (i.e. no dimensions) is drawn as a bar with height equal to $2 \cdot I_y / W_{ymin}$ and width equal to $2 \cdot I_z / W_{zmin}$.

SECTION-AS-SOLID

Draw the beam elements as solid objects, i.e. with their sections fully drawn and extruded along the beam element length. Eccentricities are taken into account. The display of eccentricities is enlarged by the SET GRAPHICS SIZE-SYMBOLS SECTION-FACTOR command.

For general section see the explanation for option OUTLINE-SECTION.

SHOW-ECCENTRICITY

This option switches ON and OFF graphic visualisation of eccentricities. The visualisation is in the form of narrow triangles (rods) between the nodes and the beam ends. This visualisation may be used independently of the other BEAM-ELEMENT settings. The display of eccentricities is enlarged by the SET GRAPHICS SIZE-SYMBOLS SECTION-FACTOR command.

SIMPLE-SECTION

Draw the beam elements with their sections in a simplified manner and taking their eccentricities into account. The display of eccentricities is enlarged by the SET GRAPHICS SIZE-SYMBOLS SECTION-FACTOR command.

For general section see the explanation for option OUTLINE-SECTION.

ELEMENT-THICKNESS

Select labelling mode for element thickness.

NUMERICAL

Thickness of 2-D elements is shown by values.

SYMBOLIC	Thickness of 2-D elements is shown graphically. In addition to the middle surface drawn with solid lines one side of the surface is drawn with dotted lines (the negative local z-side) and the other surface with broken lines (the positive local z-side). The display of thickness is enlarged by the SET GRAPHICS SIZE-SYMBOLS SECTION-FACTOR command.
FILLED-ELEMENT	Switch ON or OFF filling of 2-D and 3-D elements with a colour.
LAYERED-ELEMENTS	Select display mode for layered elements.
LAYERS	Plate layers are drawn as simple shells with their eccentricities. Stiffener layers are drawn as simple lines (through the neutral axes).
LOCAL-AXIS	Layered elements are drawn as simple shells only (eccentricities are ignored). Plate and stiffener layers are not explicitly drawn. In addition, a local axis is drawn in one of the nodes.
OUTLINE-AREA	Plate layers are drawn as two surfaces, one for the top and one for the bottom layers of the plates (with their eccentricities). Stiffener layers are drawn as simple lines (through the neutral axes) between cross sections. The stiffener cross sections are drawn both with outline of the sections as defined and scaled to represent the area corresponding to the element size. This representation of area is illustrated in Figure 5.55: If the element size exactly matches a whole number of stiffener spacings, for example 2 as in the sketch to the left, the sum of the stiffener areas corresponds exactly to the effective stiffener area. If the element size is, for example, between 2 and 3 stiffener spacings as in the sketch to the right then the areas of the 2 stiffeners drawn are scaled so as to represent the effective stiffener area. Note that the number of stiffeners drawn will be the element size divided by the stiffener spacing and truncated to a whole number. At least one stiffener will, however, always be drawn. The stiffeners are centrally positioned on the element.
OUTLINE-SECTION	Plate layers are drawn as simple shells (with their eccentricities). Stiffener layers are drawn as simple lines (through the neutral axes) between outlines of the cross sections as.
SHELL-ONLY	Layered elements are drawn as simple shells only (eccentricities are ignored). Plate and stiffener layers are not explicitly drawn.
SOLID-AREA	The plate and stiffener layers are drawn as for the OUTLINE-AREA option, see this, and in addition the stiffeners are drawn as extrusions of the cross sections, i.e. like solids.

SOLID-SECTION	The plate and stiffener layers are drawn as for the OUTLINE-AREA option, see this, except that no scaled stiffener cross sections are drawn and in addition the stiffeners are drawn as extrusions of the cross sections, i.e. like solids.
LOAD	Select presentation mode for load display.
ARROW	Load values are shown as arrows.
CONNECTED-ARROWS	Load values are shown as arrows and with their tails connected.
NUMERICAL	Load values are shown as numerical values.
MIDNODE-ELEMENTS	Select how to draw elements with mid-nodes, e.g. eight node shells elements.
NORMAL	The complete element is drawn.
SIMPLIFIED	Draw the elements as if no mid-nodes exist. This will speed up the display and may, for example, be used for a hidden mode display with many elements.

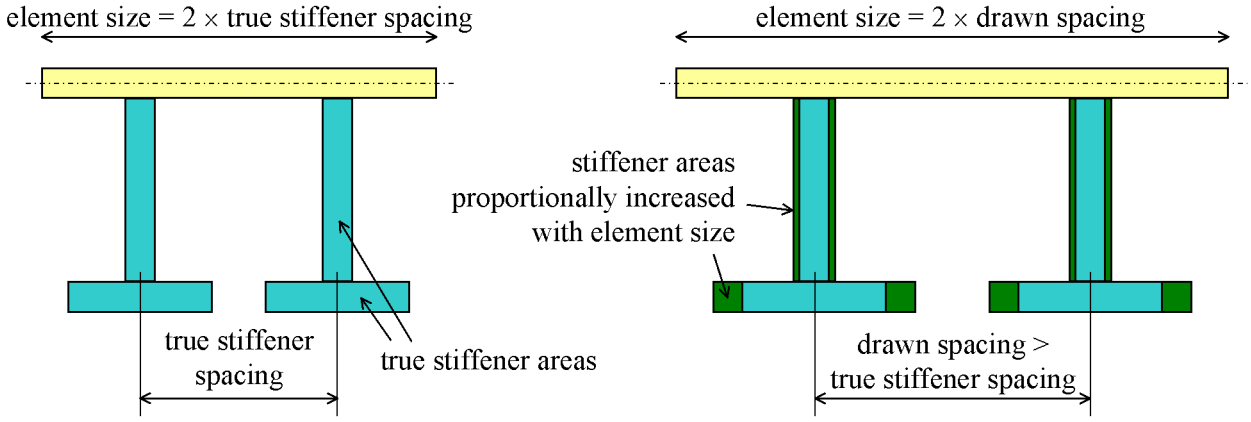


Figure 5.55 Display of layered element — SECTION option versus AREA option

SET GRAPHICS SIZE-SYMBOLS

...	SIZE-SYMBOLS	BODY-NAMES	value
		BOUNDARY-CONDITION-SYMBOLS	
		ELEMENT-NORMAL	
		ELEMENT-NUMBERS	
		ELEMENT-THICKNESS	
		ELEMENT-TYPES	
		FACTOR	
		GEOMETRY-NAMES	
		LINE-DIVISIONS	
		LINE-NAMES	
		LOAD-ARROW-SIZE	
		LOAD-VALUES	
		MATERIAL-NAMES	
		NODE-NUMBERS	
		NODE-SYMBOLS	
		ONE-NODED-ELEMENT-SYMBOLS	
		ORIGIN-SYMBOLS	
		POINT-NAMES	
		POINT-SYMBOLS	
		SECTION-FACTOR	
		SHAPE-NAMES	
		SURFACE-NAMES	
		SURFACE-NORMAL	

PURPOSE:

The command specifies (alters) the sizes of symbols appearing in the display and plot. The sizes are given in millimetres (approximately).

PARAMETERS:

BODY-NAMES Alter the sizes of body names, default is 2.

BOUNDARY-CONDITION-SYMBOLS Alter the sizes of boundary condition symbols, default is 7.5.

ELEMENT-NORMAL	Alter the sizes of 2-D element normals, default is 2.
ELEMENT-NUMBERS	Alter the sizes of element numbers, default is 2.
ELEMENT-THICKNESS	Alter the sizes of element thickness values, default is 2.
ELEMENT-TYPES	This label is presently not available.
FACTOR	Multiply all current label sizes by this factor. The factor has no influence on LOAD-ARROW-SIZE, ORIGIN-SYMBOL and SECTION-FACTOR.
GEOMETRY-NAMES	Alter the sizes of all geometry names, default is 2.
LINE-DIVISIONS	Alter the sizes of line divisions, i.e. number of elements for lines, default is 2.
LINE-NAMES	Alter the sizes of line names, default is 2.
LOAD-ARROW-SIZE	Alter the size of the maximum load arrow, default is 15.
LOAD-VALUES	Alter the sizes of load values, default is 2.
MATERIAL-NAMES	Alter the sizes of material names, default is 2.
NODE-NUMBERS	Alter the sizes of node numbers, default is 2.
NODE-SYMBOLS	Alter the sizes of node symbols (see the LABEL command), default is 1.
ONE-NODED-ELEMENT-SYMBOLS	Alter the sizes of spring, damper and mass elements, default is 2.
ORIGIN-SYMBOLS	Alter the sizes of the origin symbol, default is 10.
POINT-NAMES	Alter the sizes of point names, default is 2.
POINT-SYMBOLS	Alter the sizes of point symbols (the X), default is 1.5.
SECTION-FACTOR	Multiply symbolic representation of cross sections and element thicknesses by this factor, default is 2.
SHAPE-NAMES	This label is presently not available.
SURFACE-NAMES	Alter the sizes of surface names, default is 2.
SURFACE-NORMAL	Alter the sizes of surface normal, default is 6.
value	For most options: Give symbol size in millimetres, legal range is 1.0 - 100.0. For option FACTOR: factor used on all labels. For option SECTION-FACTOR: factor on thickness, sections and eccentricities.

SET INSIDE / OUTSIDE

...	INSIDE	select-surfaces	[corner-point]	POINT	point-name		
				COORDINATES	x	y	z
				X			
				Y			
	OUTSIDE			Z			
				-X			
				-Y			
				-Z			

PURPOSE:

The command defines the inside/outside of a surface. This definition is used in applying normal pressure loads, see Section 3.12.3.

A corner point of the surface is referred to together with a point on the inside/outside of the surface.

PARAMETERS:

INSIDE	The inside of the surface will be defined.
OUTSIDE	The outside of the surface will be defined.
select-surfaces	Select surfaces. See Section 5.1 on how to perform a selection.
corner-point	A corner point of the surface. This is not requested when more than one surface is selected.
POINT	An existing point is used for defining the inside/outside of the surface(s).
point-name	Name of a point.
COORDINATES	A point in space defined by coordinates is used for defining the inside/outside of the surface(s).
x y z	Cartesian coordinates.
X Y Z -X -Y -Z	A point in infinity along the positive/negative X, Y or Z axes is used for defining the inside/outside of the surface(s).

NOTES:

When selecting several surfaces the inside/outside definition must apply to all points of all selected surfaces, otherwise ambiguities may arise.

The last inside/outside definition for a surface is the valid one for all loads irrespective of whether the loads were defined before or after the inside/outside definition. In other words: a surface can only have one inside/outside setting.

SET JOURNALLING

...	JOURNALLING	GRAPHICS	...	ON
		PRINT		OFF

PURPOSE:

The command switches on and off logging in command log file (see Chapter 4) of print or graphics commands.

PARAMETERS:

GRAPHICS	Switch for graphics commands like DISPLAY, SET GRAPHICS EYE-DIRECTION, ROTATE, LABEL, ZOOM, etc.
PRINT	Switch for print commands.
ON	Switch on.
OFF	Switch off.

SET MAX-ELEMENT-LENGTH

...	MAX-ELEMENT-LENGTH	select-lines	size

PURPOSE:

The command defines the maximum size of elements (element edges) for a line or a group of lines.

See Section 3.4 for more information.

PARAMETERS:

select-lines	Select lines. See Section 5.1 on how to perform a selection.
size	The maximum element size

NOTES:

See also the SET NUMBEROF-ELEMENTS command.

SET MESH

...	MESH	COORDINATE-SYSTEM			select-geometry		coord-name
		CORNER-TYPE		surface-name	point-name	COLLAPSED-EDGE[a]	
						COLLAPSED-QUARTER-POINT-EDGE[b]	
						CORNER	
						CUT-CORNER	
						LARGE-CUT-CORNER	
						NOT-CORNER	
						SMALL-CUT-CORNER	
						TRIANGULAR-BOTH-NONE	
						TRIANGULAR-BOTH-ONE	
						TRIANGULAR-BOTH-TWO	
						TRIANGULAR-ELEMENT	
						TRIANGULAR-POST-NONE	
						TRIANGULAR-POST-ONE	
						TRIANGULAR-POST-TWO	
						TRIANGULAR-PRE-NONE	
						TRIANGULAR-PRE-ONE	
						TRIANGULAR-PRE-TWO	
		EDGE	RECTANGULAR		surface	line	number-of-elements
		METHOD[c]					
		VERSION		version-number			
				LATEST			

a. This option is presently inactive.
b. This option is presently inactive.
c. This option is presently inactive.

PURPOSE:

The command sets parameters controlling the creation of the mesh, especially corner types for surfaces.

See Section 3.4.4 for general information as well as examples of corner types.

PARAMETERS:

COORDINATE-SYSTEM	Set the coordinate system to use for creating the mesh. This option allows the mesh to be created in a cylindrical or spherical coordinate system. This has certain advantages for creating irregular meshes as explained in see Section 3.4.5.
select-geometry	Select appropriate geometry being bodies and/or surfaces See Section 5.1 on how to perform a selection.
coord-name	Name of a previously defined coordinate system.
CORNER-TYPE	Set a corner type for a surface.
surface-name	Name of the surface. Wild-card name is not allowed.
point-name	Name of a point on the border of surface-name.
CORNER	The point will be a mesh-corner. See the note below.
CUT-CORNER	The point will be a cut-corner.
LARGE-CUT-CORNER	The point will be a large-cut-corner.
NOT-CORNER	The point will be a not-mesh-corner.
SMALL-CUT-CORNER	The point will be a small-cut-corner.
TRIANGULAR-BOTH-NONE	See Section 3.4.4 for an explanation of this option.
TRIANGULAR-BOTH-ONE	See Section 3.4.4 for an explanation of this option.
TRIANGULAR-BOTH-TWO	See Section 3.4.4 for an explanation of this option.
TRIANGULAR-ELEMENT	See Section 3.4.4 for an explanation of this option.
TRIANGULAR-POST-NONE	See Section 3.4.4 for an explanation of this option.
TRIANGULAR-POST-ONE	See Section 3.4.4 for an explanation of this option.
TRIANGULAR-POST-TWO	See Section 3.4.4 for an explanation of this option.
TRIANGULAR-PRE-NONE	See Section 3.4.4 for an explanation of this option.
TRIANGULAR-PRE-ONE	See Section 3.4.4 for an explanation of this option.
TRIANGULAR-PRE-TWO	See Section 3.4.4 for an explanation of this option.
EDGE	See explanation for option RECTANGULAR below.
RECTANGULAR	This option allows creating a regular rectangular mesh (quadrilateral elements) for an L-shaped surface. The mesh of such a surface will otherwise be distorted. Note that the number of el-

	ements of opposite lines must be consistent with a regular rectangular mesh. See Figure 5.56.
surface	Select a surface.
line	A line of the surface. Which to select is shown in Figure 5.56. (Note that the prompt for this information is misleading.)
number-of-elements	Number of elements along a line. Which to select is shown in Figure 5.56. (Note that the prompt for this information is misleading.)
VERSION	Set version of meshing algorithm. New versions of Prefem may and may not include updates to (new version of) the meshing algorithm. This option allows using an old version of the meshing algorithm within the latest version of Prefem. This may be relevant when a command input file created for a previous Prefem version is read into the latest version.
version-number	The version number (a number: 1, 2, ...). The Status List will contain information on the meshing version number.
LATEST	The meshing version of the current Prefem version.

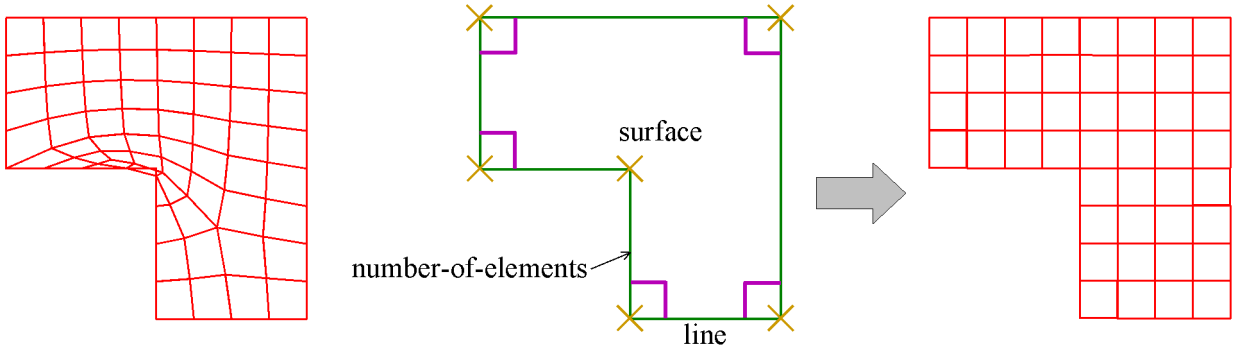


Figure 5.56 Mesh for an L-shaped surface

NOTES:

The SET MESH-CORNER and SET NOT-MESH-CORNER commands have the same effect as the corresponding SET MESH CORNER-TYPE commands.

The mesh-corners can be visualised by the LABEL MESH-CORNER command.

The SET MESH-PARAMETERS MAX-MESH-CORNER-ANGLE command allows overruling the setting of mesh-corner. See this.

SET MESH-CORNER / NOT-MESH-CORNER

...	MESH-CORNER	surface-name	point-name	*
	NOT-MESH-CORNER		END	

PURPOSE:

The command sets mesh-corners and not-mesh-corners for surfaces. The user is, however, recommended to use the SET MESH CORNER-TYPE command as this includes all alternative corner types.

See Section 3.4.4 for general information as well as examples of corner types.

PARAMETERS:

MESH-CORNER	Set mesh-corners.
NOT-MESH-CORNER	Set not-mesh-corners.
surface-name	Name of a single surface. Wild-card name is not allowed.
point-name	Name of a point on the border of surface-name to be set to mesh-corner or not-mesh-corner. Close the list of points by entering END.

NOTES:

The SET MESH-PARAMETERS MAX-MESH-CORNER-ANGLE command allows overruling the setting of mesh-corner. See this.

SET MESH-PARAMETERS

...	MESH-PARAMETERS	COORDINATE-FINENESS	FINE
			NORMAL
			COARSE
		MAX-MESH-CORNER-ANGLE	angle
		OPTIMIZE-TRIANGLES	ON
			OFF
		SOLID-ELEMENT-SHAPE	sol-shape-param
		SURFACE-ELEMENT-SHAPE	surf-shape-param

PURPOSE:

The command sets parameters for the FE mesh creation. The parameters are not stored on the model file, and are general parameters, i.e. not associated with individual surfaces or bodies. OPTIMIZE-TRIANGLES is the only parameter affecting the mesh topology. The other parameters only affect the calculation of the nodal coordinates, and only in the event of irregular element meshes.

These mesh parameters will influence the mesh created in a way that is not always easy for the user to foresee. And changing the parameters will only to a limited degree improve a bad mesh. However, the parameters may be used as follows: delete the bad mesh, change a parameter, re-create the mesh, and visually check the result.

PARAMETERS:

COORDINATE-FINENESS — Set the required degree of accuracy for the nodal coordinate calculation.

FINE NORMAL COARSE — The required degree of accuracy. FINE will require more calculation time.

MAX-MESH-CORNER-ANGLE — This option overrules the mesh-corner setting by the SET MESH CORNER-TYPE command. A maximum mesh-corner angle is set. If the mesh-corner angle is greater than the given value then the surface will be meshed with not-corner at this point irrespective of whether the point has been defined as MESH-CORNER or not.

angle — The maximum mesh-corner angle

OPTIMIZE-TRIANGLES — Determines whether pairs of triangles are optimised by splitting the quadrilateral on the shortest diagonal. Not used for solid elements.

ON / OFF — Switch the optimisation on and off.

SOLID-ELEMENT-SHAPE	Set a parameter influencing solid element shapes.
sol-shape-param	Parameter influencing the iteration used to calculate the nodal coordinates in the interior of the bodies. Legal values are between 0.0 and 0.5. Default value is 0.5.
SURFACE-ELEMENT-SHAPE	Set a parameter influencing surface element shapes.
surf-shape-param	Parameter influencing the iteration used to calculate the nodal coordinates. Legal values are between 0.0 and 1.0. Default value is 0.7.

SET NAMING

...	NAMING	CUT	DEFAULT	
			MASK	mask

PURPOSE:

The command is used for determining geometry names resulting from use of the CUT command.

When cutting geometry new geometry names will automatically be created by the program. For example, cutting a line will yield two lines of which one will be given the original name of the line and the program will determine a name for the other. By default, the program will choose names like POi for points, LIi for lines, etc., where i is a number.

Provided the GENERATE command, or at least the naming system of that command, has been used the SET NAMING command opens for maintaining this system for new geometry names. See the GENERATE command for information on this naming system.

For example, if there exists a line between the two points AP11 and AP13 and this line is cut then a new point is created. The SET NAMING CUT command allows this point to be named AP12 rather than the default name POi.

Another example: The line between the two points AP11 and AP12 is cut. In this case there is no space in the numbering system as in the previous example. This is solved by employing an extended numbering system:

0 A 1 B 2 C 3 D 4 E 5 F 6 G 7 H 8 I 9 J

The SET NAMING CUT command allows the new point to be named AP1B rather than the default name POi. (If a line between the two points AP11 and AP1B is cut then a name following this extended system cannot be established because there is no space in the numbering system, a default name (POi) is used in such a case.)

A so-called mask name is specified by the user to force new names to follow the naming system. The mask name for the two examples above will be:

&&IJ

where the first '&' represents the 'id' A and the second represents P for points (and I, J and K for lines, S, T, U for surfaces, B for bodies). The 'IJ' represents a single digit for each of the topological I and J directions.

The mask name:

&&IIJJKK

will be suitable for specifying maintenance of the naming system when cutting geometry consisting of names including two digits for each of the three topological directions.

The mask name:

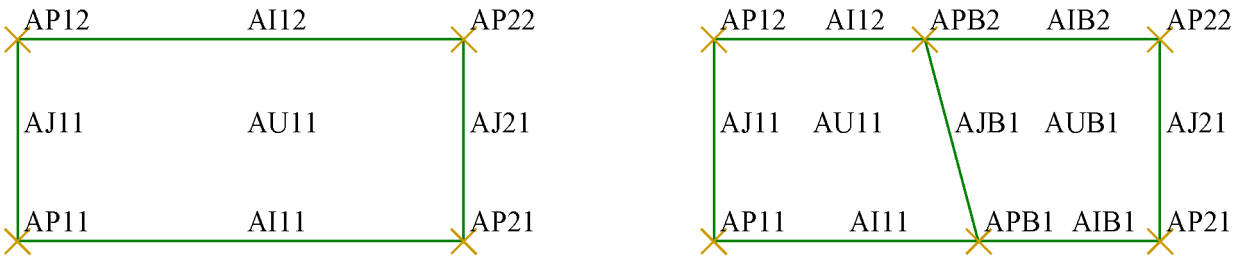
&&&IJKK

will be suitable for names including an ‘id’ of two characters (corresponding to the two first ‘&’s) and one digit for each of the topological directions I and J and two digits for the topological direction K.

To determine a new name there will be an interpolation in the extended numbering system above and the middle ‘value’ is used. Examples: in between 2 and 4, 3 is taken; in between 2 and 6, 4 is taken; in between 5 and 6, F is taken.

All geometry names (points, lines, surfaces and bodies) matching the mask name will, when cut, result in new names following the system.

The example of Figure 5.57 illustrates its use.



The surface AU11 to the left is cut and new points, lines and surfaces are created and given names. The mask name &&IJ has been specified. New point names will be as shown to the right. Line and surface names follow the new point names.

Figure 5.57 Naming geometry after cutting

If a geometry name does not adhere to the naming system, i.e. the mask name does not fit, then a default name (POi for points, LIi for lines, etc.) will be used for the new geometry.

PARAMETERS:

CUT	Determine geometry names in connection with CUT command.
DEFAULT	By default all names are generated by incrementing the number ‘i’ in the standard names: POi for points, LIi for lines, SUi for surfaces and BOi for bodies.
MASK	Use a mask type of naming.
mask	The mask name.

SET NOT-MESH-CORNER

...	NOT-MESH-CORNER	surface-name	point-name	*
			END	

PURPOSE:

The command sets not-mesh-corners.

See the SET MESH-CORNER command for a full description.

SET NUMBEROF-ELEMENTS

...	NUMBEROF-ELEMENTS	select-lines	nelm

PURPOSE:

The command defines the number of elements for a line or a group of lines.

See Section 3.4 for more information.

PARAMETERS:

select-lines	Select lines. See Section 5.1 on how to perform a selection.
nelm	Number of elements for the selected lines.

NOTES:

See also the SET MAX-ELEMENT-LENGTH command.

SET OUTSIDE

...	OUTSIDE	select-surfaces	[corner-point]	POINT	point-name		
				COORDINATES	x	y	z
				X			
				Y			
				Z			
				-X			
				-Y			
				-Z			

PURPOSE:

The command defines outside of surfaces for the benefit of load application.

See the SET INSIDE command for a full description.

SET PLOT

...	PLOT	COLOUR	ON	
			OFF	
		FILE	file-prefix	file-name
		FORMAT	number	
			CGM-BINARY	
			HGPL-2	
			HPGL-7550	
			POSTSCRIPT	
			SESAM-NEUTRAL	
			WINDOWS-PRINTER	
		ORIENTATION	PORTRAIT	
		PAGE-SIZE	A4	

PURPOSE:

The command sets parameters for plotting. The settings must be done prior to the PLOT command.

PARAMETERS:

COLOUR — Switch ON or OFF colours. The default is OFF. Colours are only supported by CGM-BINARY, POSTSCRIPT and HPGL-2. Give this command after choosing format and before saving the plot file.

FILE — Set file prefix and name for the plot file. Default is the same as for the model (and command log) file. The file extension depends on the type of format.

file-prefix file-name — The file prefix and name for the plot file.

FORMAT — Set the plot format. See Section 4.1.6 for information on which format to choose.

number — A number corresponding to the plot format may alternatively be given but you will normally not know this.

CGM-BINARY — The CGM (Computer Graphics Metafile) is chosen. File extension is .CGM.

HPGL-2 — A Hewlett Packard plot format. File extension is .HPG2.

HPGL-7550 — A Hewlett Packard plot format. File extension is .HP70.

POSTSCRIPT — The PostScript plot format. File extension is .PS.

SESAM-NEUTRAL	A plot format of the SESAM system. File extension is .PLO.
WINDOWS-PRINTER	The plot will be sent directly to the on-line printer. No plot file will be generated. This is the default format on PC.
ORIENTATION	Set the page orientation. Valid for PostScript format only.
PORTRAIT	Portrait orientation. This is default.
PAGE-SIZE	Set the plot page size. All sizes are not available for all plot formats. For SESAM-NEUTRAL this setting is irrelevant as the page size is set within the PLOT command. Valid for PostScript format only.
A4	European standard page sizes (paper formats). See explanation for the PLOT command.

SET PREFIX-NAME

...	PREFIX-NAME	ON	prefix
		OFF	

PURPOSE:

The command sets a prefix for geometry names, i.e. a string that is put in front of subsequently generated geometry names.

The command allows setting various prefixes during the modelling and in this way ease later identification and selection (by wild-card) of geometry.

The structure of the geometry names resulting from this command is:

prefix + geometry type + number

where ‘geometry type’ is:

- P for points
- L for lines
- S for surfaces
- B for bodies

‘number’ is a number incremented by the program.

PARAMETERS:

prefix The prefix string

NOTES:

Note that a geometry name is limited to eight characters, the prefix should, therefore, be limited to a few characters, say four, in order to avoid any problems for geometry names with high numbers.

Exiting and re-entering the program will not affect the prefix-setting.

SET PRINT

...	PRINT	ALPHABETIC	ON			
			OFF			
		COORDINATE-SYSTEM	CARTESIAN			
			CYLINDRICAL	X-AXIS		
				Y-AXIS		
				Z-AXIS		
		DESTINATION	FILE			
			SCREEN			
		FILE	file-prefix	file-name		
		LOAD	ACCUMULATION	ON		
				OFF		
			EXTENT	ALL		
				GEOMETRY		
				MESH		
			NODE-NUMBER	4-DIGITS		
				5-DIGITS		
			TEMPERATURE	DIFFERENCE-CHECK	...	ON
				MIDSIDE-NODE		OFF
				NODAL-FROM-SOLID		

PURPOSE:

The command sets parameters for print.

PARAMETERS:

ALPHABETIC — Switch alphabetic sorting in tables over geometry ON and OFF.

COORDINATE-SYSTEM — Set coordinate system for output of nodal coordinates.

CARTESIAN — Use the cartesian coordinate system of the model. This is the default choice.

CYLINDRICAL — Use a cylindrical coordinate system with either of the cartesian axes used as the cylinder axis. I.e. all nodal coordinates are for the purpose of printed output (only!) transformed from the car-

	tesian coordinate system of the model to a cylindrical r-ϕ-z system.
X-AXIS	The cartesian X-axis is used as the cylinder axis. The cartesian Y-axis defines the ϕ=0 plane.
Y-AXIS	The cartesian Y-axis is used as the cylinder axis. The cartesian Z-axis defines the ϕ=0 plane.
Z-AXIS	The cartesian Z-axis is used as the cylinder axis. The cartesian X-axis defines the ϕ=0 plane.
DESTINATION	Select FILE or SCREEN for output from subsequent PRINT commands. A PRINT ALL command will always send its print to FILE. Default for all other PRINT commands is SCREEN.
FILE	Set file prefix and name for the print file. Default is the same as for the model file. The file extension is .LIS.
LOAD	Set format of print of loads.
ACCUMULATION	As the same node or element may be given the same type of load for the same load case more than once (the resulting load is an accumulation of all these loads) there is a possibility for accumulating these loads in the print tables. This accumulation may be switched ON and OFF. Default is off. This is only relevant for printing of loads for elements and nodes (SET PRINT EXTENT MESH).
EXTENT	Select printing of loads to be in the form of tables over loads for geometric entities (GEOMETRY) or tables over loads for elements and nodes (MESH) or for both geometric entities and elements/nodes (ALL). GEOMETRY is the default choice.
NODE-NUMBER	Set number of digits for the node numbers to 4-DIGITS or 5-DIGITS. 4 digits makes the print tables easier to read and is the default choice. Only use 5 digits when necessary.
TEMPERATURE	Set format for printing temperature loads.
NODAL-FROM-SOLID	For use when nodal temperature values are printed. Only the temperature applied to the first element connected to the node is printed. If requested the program will give a message if different temperatures have been applied to the node (see the DIFFERENCE-CHECK option). Only nodes connected to elements where temperature loads have been applied are printed. Only solid elements are printed and checked. This command translates the internal representation of temperature loads as element loads to nodal loads.
MIDSIDE-NODE	As temperature loads in mid-side nodes are not taken into account in the analysis printing of these may be switched off.

DIFFERENCE-CHECK	This option is relevant only in combination with the NODAL-FROM-SOLID switched ON. A question mark (?) is printed after the node number if the node has been given different temperatures in different elements connected to the node.

SET PROJECTION

...	PROJECTION	surface-name	shape-name

PURPOSE:

The command determines that nodes created at the *interior* of a surface shall be projected onto a shape. This projection is performed during the meshing operation. See Section 3.4.5 for information on mesh for surfaces.

PARAMETERS:

surface-name The name of a surface.

shape-name The name of a previously defined shape, see the DEFINE SHAPE command.

NOTES:

Alternatively to using this command the DEFINE SURFACE command may have defined such projection, see this.

SET TASK

...	TASK	ELEMENT-TYPE	FRAME	...	ON
			SHELL		OFF
			SOLID		ONLY
		FULL-MENUS			
		SHORT-MENU-MODE			
		LONG-MENU-MODE			
		MODELLING	GEOMETRY	...	ON
			PROPERTY		OFF
			LOAD		ONLY
			MESH		

PURPOSE:

This command enables suppressing commands irrelevant for the current task.

The two filters, MODELLING and ELEMENT-TYPE, can be applied at the same time. These filters switches on and off commands related to various modelling features. The SHORT-MENU-MODE and LONG-MENU-MODE are qualifiers that can be selected in addition to the filtering.

PARAMETERS:

ELEMENT-TYPE	Filter for element types.
FRAME	Switch for frame type elements (1-D elements).
SHELL	Switch for shell type elements (2-D elements). This switch also encompasses 1-D elements.
SOLID	Switch for solid type elements (3-D elements). This switch also encompasses 1-D elements.
ON	Switch on the selected modelling feature.
OFF	Switch off the selected modelling feature.
ONLY	Switch on the selected modelling feature and switch the other features off.
FULL-MENUS	Suppress no commands at all. As if LONG-MENU-MODE is selected and all switches under the filters are switched on.
SHORT-MENU-MODE	Suppress less used commands.

LONG-MENU-MODE	Suppress no commands within the limits of the selected filters.
MODELLING	Filter for modelling features.
GEOMETRY	Switch for geometry modelling.
PROPERTY	Switch for property modelling.
LOAD	Switch for load modelling.
MESH	Switch for mesh modelling.

EXAMPLES:

The example below will suppress modelling features not related to geometry modelling of a frame and shell structure. The SHORT-MENU-MODE qualifier makes only the most commonly used commands available.

```
SET TASK SHORT-MENU-MODE
         MODELLING GEOMETRY ONLY
         ELEMENT-TYPE-FILTER SOLID OFF
         END
         END
```

SET TOLERANCE

...	TOLERANCE	COORDINATES	coord-tol
		LOAD-DIFFERENCE	load-tol
		SNAP	snap-tol
		UNIT-VECTOR	unit-vect-tol

PURPOSE:

The command sets tolerance values.

PARAMETERS:

COORDINATES — Set the tolerance value for geometric operations. The coordinate tolerance value is used to check geometrical coincidence between coordinates, e.g. when defining new points by the COPY command. The tolerance should be set in accordance with the units used (e.g. m or mm?) and the minimum distances between points.

coord-tol — Coordinate tolerance value in the same units as the coordinates. The default value is 0.1.

LOAD-DIFFERENCE — Set the tolerance for print of nodal values of loads. The tolerance is used to check whether two elements have the same temperature load value in nodes that the elements have in common.The default value is 0.0.

load-tol — Tolerance used in load printing.

SNAP — Set the tolerance for snapping to existing points when using the graphic feature for cutting (within the CUT command). The snap tolerance is given in *millimetres as measured on the screen*. (The snap tolerance is automatically converted to model units depending on the current zoom factor.) Note that if the converted snap tolerance becomes smaller than the coordinate tolerance then the latter will be used.

snap-tol — Snap tolerance for graphic cutting. The default value is 2.0.

UNIT-VECTOR — Set a tolerance used for checking whether two vectors are parallel. The cross product is checked.

unit-vect-tol — Vector parallel check tolerance, the default value is 0.001.

SET WRITE-MODE

...	WRITE-MODE	2DIMENSIONAL		
		3DIMENSIONAL		
		INCLUDE-SET	set-name	ALL
				NODES
				ELEMENTS
				NONE

PURPOSE:

The command sets the mode for writing the Input Interface File.

The user has the option to generate the Input Interface File describing the FE model either as a 2-D or a 3-D model. The SET WRITE-MODE command enables the user to specify which type of model file is to be generated. For the 2-D model the degrees of freedom corresponding to the translation in Y-direction and the rotations about the X- and Z-axes are fixed. A 2-D model in the X-Z-plane is thus generated easily and directly.

The command allows control over the storing of sets (see the DEFINE SET command) on the Input Interface File. By default, all sets (if any are defined) with their full contents (i.e. nodes and elements) are stored on the Input Interface File.

PARAMETERS:

2DIMENSIONAL	2-D model is to be stored.
3DIMENSIONAL	3-D model is to be stored, this is the default mode.
INCLUDE-SET	Control storing of a set on the Input Interface File.
set-name	Name of a previously defined set.
ALL	Both nodes and elements are stored.
NODES	Only nodes are stored.
ELEMENTS	Only elements are stored.
NONE	Neither nodes nor elements are stored.

WRITE

WRITE	superelement-number

PURPOSE:

The command writes the Input Interface File.

Note: Normally the writing of the Input Interface File is controlled by Manager, see Section 2.13.1.

ZOOM

ZOOM	FRAME	
	IN	*define zoom area*
	OUT	
	OFF	
	REDISPLAY-OFF	

PURPOSE:

The command zooms the display in and out.

PARAMETERS:

FRAME — The currently displayed part of the model will fill the display area.

IN — Zoom in by defining the area to magnify.

OUT — Zoom out by defining the area into which the current display will be fitted.

define zoom area — Use the mouse to define two diagonal points of the rectangular zoom area. This may be done in two ways:

(1) Click twice, once for each diagonal point.

(2) Press and hold in the first point, drag and release in the second point. A rubber-band will appear indicating the zoom area.

OFF — The whole model — not only the currently displayed part as is the case for the FRAME option — will fill the display area. The model will, however, not be re-displayed, i.e. the screen goes blank.

REDISPLAY-OFF — The same as the OFF option and in addition the model is displayed.

# #

#	ALL
	number-of-commands

PURPOSE:

The command initiates reading and execution of commands from a previously defined command input file. See the SET COMMAND-INPUT-FILE command.

The program will read and execute commands until either:

- An end-of-file mark is detected,
- The symbol '#' is encountered or
- The requested number of commands have been read.

By giving number-of-commands = 1 a single command will be read. The command prompt '>' will appear thereafter indicating that the following commands may be read and executed one by one by hitting carriage return. This feature may prove useful for stepping through a command input file. At any time the reading can be stopped by entering '#'.

PARAMETERS:

ALL All commands on the command input file are read and executed.

number-of-commands Number of commands to be read and executed.

APPENDIX A TUTORIAL EXAMPLES

The following tutorial examples are presented:

- Midship section
- Cylindrical tank with flat bottom and spherical top

A 1 Midship Section

The line-mode input required to create the model of Section 3.2 is presented below.

```
% ========================================================================
% Prefem input for modelling midship section
% ========================================================================
%
% --- Set naming of geometry to be done automatically
 SET DEFAULT AUTOMATIC-NAMING ON
 ..
% --- Define four points
 DEFINE POINT
   0  0  0
   0 40  0
   0 40 50
   0  0 50
 ..
% --- Define surface that will be the bulkhead
 DEFINE SURFACE PO1 PO2 PO3 PO0 PO1
 ..
% --- Cut surface first horizontally and then vertically
 CUT ALL-SURFACES-INCLUDED PREDEFINED-PLANE XY-PLANE 18
 CUT ALL-SURFACES-INCLUDED PREDEFINED-PLANE XZ-PLANE 15
% --- Round off corner to form the bilge
 DEFINE ROUNDED-CORNER LI3 LI5 10 CORNER
 ..
```

```
% --- Move point down to form sloping outer deck
 CHANGE POINT PO3 < DZ -2 >
 ..
% --- Extrude (and copy) the bulkhead to form the complete midship section
 EXTRUDE ( SU3 SU6 SU8 ) COP ( LI1 LI7 LI2 LI5 LI16 LI12 LI10 ) EXT
   GLOBAL  2 15 0 0 18 0 0 LINE-TO-SURFACE
% --- Assign thickness to surfaces
 PROPERTY THICKNESS
   ALL-SURFACES-INCLUDED 0.03
   ( EXTS12 EXTS13 EXTS22 EXTS23 ) 0.02
 ..
% --- Define and assign material to surfaces (plates) and lines (beams)
 PROPERTY MATERIAL STEEL ELASTIC 2.1E11 0.3 7850.0 0.0 0
 ..
 CONNECT MATERIAL STEEL ( ALL-SURFACES-INCLUDED ALL-LINES-INCLUDED )
 ..
% --- Set type of element: 4 node shell for all surfaces and 2 node beam for
%     selected lines (where there should be girders)
 SET ELEMENT-TYPE
   SURFACE ALL-SURFACES-INCLUDED SHELL-4NODES
   LINE ( EXTL14 EXTL24 EXTL15 EXTL25 EXTL17 EXTL27 ) BEAM-2NODES
 ..
% --- Define and assign beam cross section
 PROPERTY SECTION GIRDER I 2.0 0.8 0.04 0.02 0.8 0.04 1.0 1.0
 ..
 CONNECT SECTION GIRDER ALL-LINES-INCLUDED
 ..
% --- Specify eccentricity (the girders flush with the plates)
 PROPERTY ECCENTRICITY-BEAM
   ( EXTL24 EXTL14 EXTL25 EXTL15 ) CALCULATED-NEGATIVE-Z-OFFSET
   ( EXTL27 EXTL17 ) CALCULATED-POSITIVE-Z-OFFSET
 ..
% --- Set maximum element length for all surfaces
 SET MAX-ELEMENT-LENGTH ALL-SURFACES-INCLUDED 4
 ..
% --- Create a FE mesh for all lines and surfaces
 MESH ALL
% --- Set the mode in which a change in mesh involves re-meshing
 SET DEFAULT ADJUST-MESH ON END
 ..
% --- Adjust number of elements for the three short horizontal lines
%     next to the blige
 SET NUMBEROF-ELEMENTS ( LI12 COPL16 COPL26 ) 1
 ..
% --- Define boundary conditions (fixations)
 PROPERTY BOUNDARY-CONDITION
% --- A) Boundary conditons for the three lines/curves
   ( LI1 LI7 LI2 LI5 LI16 LI12 LI10 COPL22 COPL23 COPL20 COPL24 COPL21
   COPL26 COPL25 )
   FIX FREE FREE FREE FIX FIX
   GLOBAL
% --- B) Boundary conditons for the vertical line in the bulkhead midship
```

```
   ( LI4 LI0 )
   FREE FIX FREE FIX FREE FIX
   GLOBAL
% --- C) Boundary conditons for the end-points of the centre-line girder
   ( COPP26 PO1 )
   FIX FIX FREE FIX FIX FIX
   GLOBAL
% --- D) Boundary conditons for two points in the deck centre-line
   ( COPP25 PO2 )
   FIX FIX FIX FIX FIX FIX
   GLOBAL
 ..
% --- Define two load cases
 PROPERTY
   LOAD 1 COMPONENT-PRESSURE ( EXTS13 EXTS23 EXTS12 EXTS22 )
   GLOBAL 0.0 0.0 -500.0 END MIDDLE-SURFACE
   END
   LOAD 2 NORMAL-PRESSURE ( EXTS10 EXTS14 EXTS20 EXTS24 EXTS11
   EXTS21 EXTS15 EXTS16 EXTS25 EXTS26 )
   LINEAR-2POINTS-VARYING PO3 0.0 PO10 380.0 END MIDDLE-SURFACE
   END
 ..
% --- Delete the current mesh
 DELETE MESH ALL
 ..
% --- For selected surfaces: Change the direction of the load by changing
%     the "rotation" of the surfaces
 CHANGE ROTATION-OF-SURFACE ( EXTS21 EXTS11 )
 ..
% --- Re-create the mesh
 MESH ALL
%
% ============================================================================
% The model is now complete
```

A 2 Cylindrical Tank with Flat Bottom and Spherical Top

This example shows how to use the GENERATE command to model a cylindrical tank with flat bottom and spherical top. Three GENERATE commands are used, the last two adds to the former ones without duplicating existing geometry. The model is shown in Figure A.1.

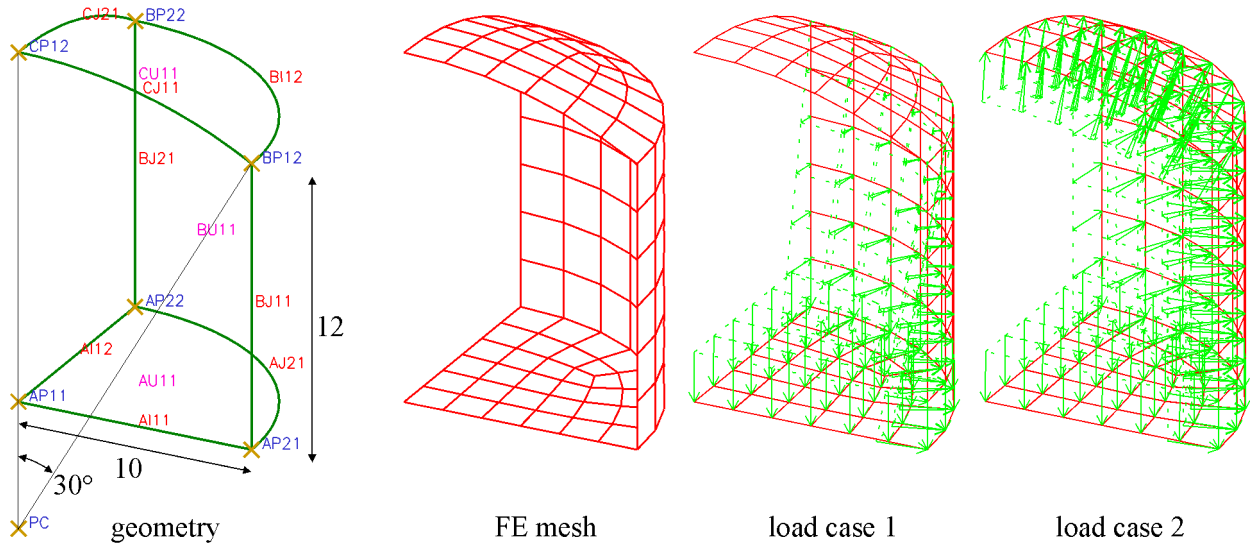


Figure A.1 Display of geometry with point, line and surface names plus FE mesh plus loads

```
% ========================================================================
% Prefem input for modelling quarter of a cylindrical tank with spherical top
% ========================================================================
%
% --- Use GENERATE to create the geometry of the flat bottom
 GENERATE SURFACE A 1 2 1 6 1 2 1 8 END
   CYLINDRICAL
   0 0 0 0 0 1 1 0 0
   0 0 0
   10 0 0 END 0 90 0 END
% --- Use GENERATE to create the geometry of the cylindrical tank
 GENERATE SURFACE B 1 2 1 8 1 2 1 6 END
   CYLINDRICAL
   0 0 0 0 0 1 1 0 0
   10 0 0
   0 90 0 END 0 0 12 END
% --- Use GENERATE to create the geometry of the spherical top
 GENERATE SURFACE C 1 2 1 8 1 2 1 6 END
   SPHERICAL
   0 0 -5.3205 0 0 1 1 0 0
   20 0 60
   0 90 0 END 0 0 30 END
```

```
% --- Give surface thicknesses, the thickness of the cylinder varies
 PROPERTY THICKNESS
   A* 0.05
   B* LINEAR-2POINTS-VARYING AP21 0.05 BP12 0.01
   C* 0.01
 ..
% --- Set type of element
 SET ELEM-TYPE SURF ALL-SURFACES-INCL SHELL-4NODES
 ..
% --- Define a shape and project the mesh of the spherical top onto it
 DEFINE POINT PC 0.0 0.0 -5.3205
 ..
 DEFINE SHAPE SPHERE SPH1 PC 20
 ..
 SET PROJECTION C* SPH1
 ..
% --- Define boundary conditions
 PROPERTY BOUNDARY-CONDITION
   ( AI11 BJ11 CJ11 )
   FREE FIX FREE FIX FREE FIX
   GLOBAL
   ( AI12 BJ21 CJ21 )
   FIX FREE FREE FREE FIX FIX
   GLOBAL
   AP11
   FIX FIX FREE FIX FIX FIX
   GLOBAL
   AP21
   FREE FIX FIX FIX FREE FIX
   GLOBAL
   AP22
   FIX FREE FIX FREE FIX FIX
   GLOBAL
   CP12
   FIX FIX FREE FIX FIX FIX
   GLOBAL
 ..
% --- Define and assign material to surfaces
 PROPERTY MATERIAL STEEL ELASTIC 2.1E11 0.3 7850.0 0.0 0
 ..
 CONNECT MATERIAL STEEL ALL-SURFACES-INCLUDED
 ..
% --- Define loads:
%     case 1 is inside pressure from liquid up to top of cylinder
%     case 2 is nside pressure from gas
 PROPERTY
   LOAD 1 NORMAL-PRESSURE ( A* B* )
   LINEAR-2POINTS-VARYING AP21 1000. BP12 0.0 END MIDDLE-SURFACE
   END
   LOAD 2 NORMAL-PRESSURE ALL-SURFACES-INCLUDED 2000. END MIDDLE-SURFACE
   END
 ..
```

```
% --- Change "rotation" of cylinder and top surfaces to change surface normal
%     thereby making normal pressure act in proper direction
 CHANGE ROTATION-OF-SURFACE ( B* C* )
 ..
% --- Create the mesh
 MESH ALL
% ============================================================================
% The model is now complete
```

APPENDIX B THEORY

B 1 Formulae for Sectional Parameters

The formulae employed in Prefem for computing the sectional parameters for the various beam cross sections are given in the following. The formulae are taken from Ref. /1/, Ref. /2/ and Ref. /3/.

The following notation is used:

AREA	Cross sectional area
IX	Torsional moment of inertia about shear centre
IY	Moment of inertia about y-axis
IZ	Moment of inertia about z-axis
IYZ	Product of inertia about y- and z-axes
WXMIN	Minimum torsional sectional modulus about shear centre
WYMIN	Minimum sectional modulus about y-axis
WZMIN	Minimum sectional modulus about z-axis
SHARY	Shear area in the direction of y-axis
SHARZ	Shear area in the direction of z-axis
SHCENY	Shear centre location y-component
SHCENZ	Shear centre location z-component
SY	Static area moment about y-axis
SZ	Static area moment about z-axis

CY Centroid location from bottom right corner y-component

CZ Centroid location from bottom right corner z-component

Variables other than the ones above are only temporary.

Note: The local x-axis of the beam or truss element goes through the centroid of the cross section. I.e. the nodal displacements and consequently the cross sectional constants above refer to this axis. The torsional moment of inertia, however, refers to the shear centre. In most beam element theories the torsional d.o.f. is not coupled to the transverse d.o.f.s. Therefore, when torsion is of importance the shear centre should not be located far away from the centroid of the cross section, i.e. avoid heavily unsymmetrical cross sections.

B 1.1 Bar section

B 1.1.1 Sectional Dimensions

HZ	Height
BB	Width at bottom
BT	Width at top
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

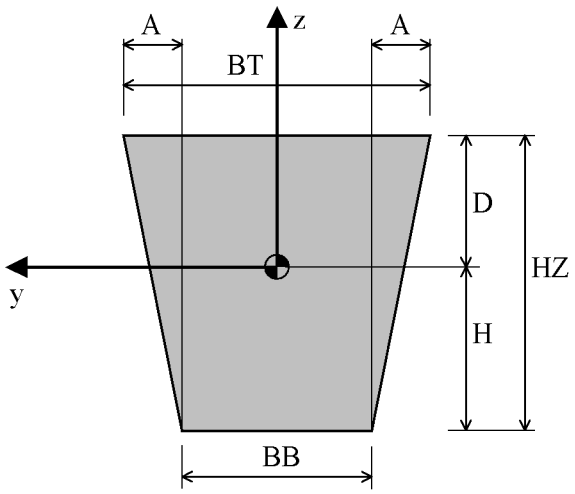


Figure B.1 Bar section

B 1.1.2 Sectional Parameters Computed

The expressions below for *IX*, *IY*, *IZ*, *WXMIN*, *WYMIN*, *WZMIN*, *SHARY* and *SHARZ* are taken from Ref. /1/. The expressions for *SHCENY* and *SHCENZ* are taken from Ref. /3/.

$$A = (BT - BB)/2$$

$$H = HZ(BB + 4A/3)/(BB + BT)$$

$$D = HZ - H$$

$$B = BT/2 - A \cdot D/HZ$$

$$BM = 2B \cdot HZ^2/(HZ^2 + A^2)$$

$$AREA = (BB + BT)HZ/2$$

$$IY = BB \cdot HZ^3/12 + HZ \cdot BB(HZ/2 - H)^2 + 2A \cdot HZ^3/36 + A \cdot HZ(2HZ/3 - H)^2$$

$$WYMIN = IY/\mathrm{MAX}(H, D)$$

$$IZ = HZ \cdot BB^3/12 + HZ \cdot A^3/18 + A \cdot HZ(BB/2 + A/3)^2$$

$$WZMIN = 2IZ/\mathrm{MAX}(BT, BB)$$

$$SY = BB \cdot H^2/2 + (B - BB/2)H^2/3$$

$$SZ = HZ(BB^2/8 + A(BB/4 + A/6))$$

$$SHARY = IZ \cdot HZ \cdot SFY/SZ$$

$$SHARZ = 2IY \cdot B \cdot SFZ/SY$$

$$SHCENY = 0$$

$$IYZ = 0$$

If $(HZ = BM)$ then
$$\begin{cases} CA = 0.141 \\ CB = 0.208 \\ IX = CA \cdot HZ^4 \\ WXMIN = CB \cdot HZ^3 \end{cases}$$

Else if $(HZ < BM)$ then
$$\begin{cases} CN = BM/HZ \\ CA = (1 - 0.63/CN + 0.052/CN^5)/3 \\ CB = CA/(1 - 0.63/(1 + CN^3)) \\ IX = CA \cdot BM \cdot HZ^3 \\ WXMIN = CB \cdot BM \cdot HZ^2 \end{cases}$$

Else
$$\begin{cases} CN = HZ/BM \\ CA = (1 - 0.63/CN + 0.052/CN^5)/3 \\ CB = CA/(1 - 0.63/(1 + CN^3)) \\ IX = CA \cdot HZ \cdot BM^3 \\ WXMIN = CB \cdot HZ \cdot BM^2 \end{cases}$$

$$SHCENZ = 0.354HZ(BT–BB)/(BT + BB) + HZ/2 - H$$

$$CY = BB/2$$

$$CZ = H$$

B 1.2 Box section

B 1.2.1 Sectional Dimensions

HZ	Height
BY	Width
TT	Thickness of top flange
TY	Thickness of webs (vertical walls)
TB	Thickness of bottom flange
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

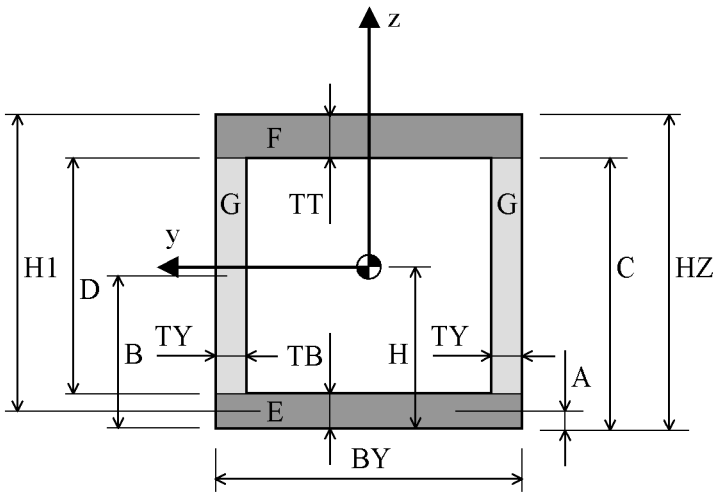


Figure B.2 Box section

B 1.2.2 Sectional Parameters Computed

The expressions below for *IX*, *IY*, *IZ*, *WXMIN*, *WYMIN*, *WZMIN*, *SHARY*, *SHARZ* and SHCENY are taken from Ref. /1/. The expression for *SHCENZ* is taken from Ref. /2/.

$$A = TB/2$$

$$B = (HZ + TB - TT)/2$$

$$C = HZ - TT/2$$

$$D = HZ - TB - TT$$

$$E = BY \cdot TB$$

$$F = BY \cdot TT$$

$$G = TY \cdot D$$

$$AREA = E + F + 2G$$

$$H = (E \cdot A + F \cdot C + 2B \cdot G)/AREA$$

$$HA = HZ - (TB + TT)/2$$

$$HB = BY - TY$$

$$IX = 4(HA \cdot HB)^2/(HB/TB + HB/TT + 2HA/TY)$$

$$IY = (BY(TB^3 + TT^3) + 2TY \cdot D^3)/12 + E(H - A)^2 + F(C - H)^2 + 2G(B - H)^2$$

$$IZ = ((TB + TT) \cdot BY^3 + 2D \cdot TY^3)/12 + G \cdot HB^2/2$$

$$IYZ = 0$$

$$WXMIN = IX(HB + HA)/(HA \cdot HB)$$

$$WYMIN = IY/\mathrm{MAX}(HZ - H, H)$$

$$WZMIN = 2IZ/BY$$

$$SY = E(H - A) + (H - TB)^2 TY$$

$$SZ = (TB + TT)BY^2/8 + G \cdot HB/2$$

$$SHARY = (IZ/SZ) \cdot (TB + TT) \cdot SFY$$

$$SHARZ = (IY/SY) \cdot 2TY \cdot SFZ$$

$$SHCENY = 0$$

$$SHCENZ = C - H - TB \cdot HA/(TB + TT)$$

$$CY = BY/2$$

$$CZ = H$$

B 1.3 Channel section

B 1.3.1 Sectional Dimensions

HZ	Height
BY	Width of top and bottom flanges
TZ	Thickness of top and bottom flanges
TY	Thickness of web
SFY	Shear factor y-direction
SFZ	Shear factor z-direction
POSWEB	=1 for web location in positive y-direction, otherwise =-1

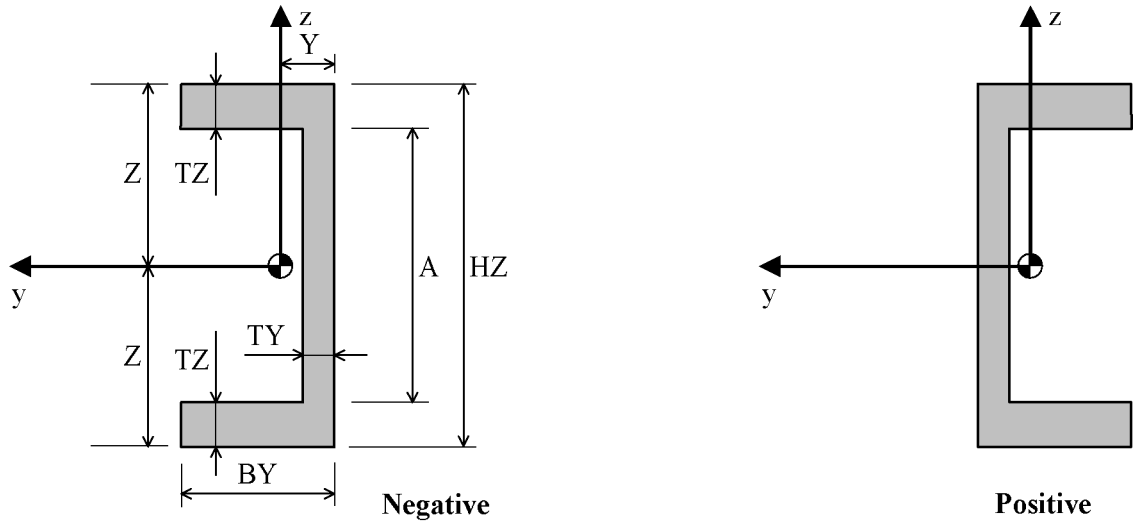


Figure B.3 Channel section

B 1.3.2 Sectional Parameters Computed

The expressions below for *IX*, *IY*, *IZ*, *WXMIN*, *WYMIN*, *WZMIN*, *SHARY* and *SHARZ* are taken from Ref. /1/. The expressions for SHCENY and *SHCENZ* are taken from Ref. /2/.

$$A = HZ - 2TZ$$

$$AREA = 2BY \cdot TZ + A \cdot TY$$

$$Y = (2TZ \cdot BY^2 + A \cdot TY^2)/(2AREA)$$

$$Z = HZ/2$$

If $(TZ = TY)$ then $\begin{cases} IX = TY^3(2BY + A - 2.6TY)/3 \\ WXMIN = IX/TY \end{cases}$

Else $\begin{cases} IX = 1.12(2BY \cdot TZ^3 + A \cdot TY^3)/3 \\ WXMIN = IX/\mathrm{MAX}(TZ, TY) \end{cases}$

$IY = TY \cdot A^3/12 + 2(BY \cdot TZ^3/12 + BY \cdot TZ \cdot ((A + TZ)/2)^2)$

$IZ = 2(TZ \cdot BY^3/12 + TZ \cdot BY \cdot (BY/2 - Y)^2) + A \cdot TY^3/12 + A \cdot TY(Y - TY/2)^2$

$IYZ = 0$

$WYMIN = 2IY/HZ$

$WZMIN = IZ/\mathrm{MAX}(BY - Y, Y)$

$SY = BY \cdot TZ(TZ + A)/2 + TY \cdot A^2/8$

$SZ = TZ(BY - Y)^2$

$SHARY = (IZ/SZ) \cdot (2TZ) \cdot SFY$

$SHARZ = (IY/SY) \cdot TY \cdot SFZ$

If $(TZ = TY)$ then $Q = (B - TY/2)^2(HZ - TZ)^2 TZ/4IY$

Else $Q = (BY - TY/2)^2 TZ/(2(BY - TY/2)TZ + (HZ - TZ)TY/3)$

If web located in positive y then $\begin{cases} SHCENY = Y - TY/2 + Q \\ CY = BY - Y \end{cases}$

Else $\begin{cases} SHCENY = -(Y - TY/2 + Q) \\ CY = Y \end{cases}$

$CZ = Z$

B 1.4 Double-bottom section

B 1.4.1 Sectional Dimensions

HZ	Height
TY	Thickness of web
TB	Thickness of bottom plate
TT	Thickness of top plate
BY	Effective width of plates
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

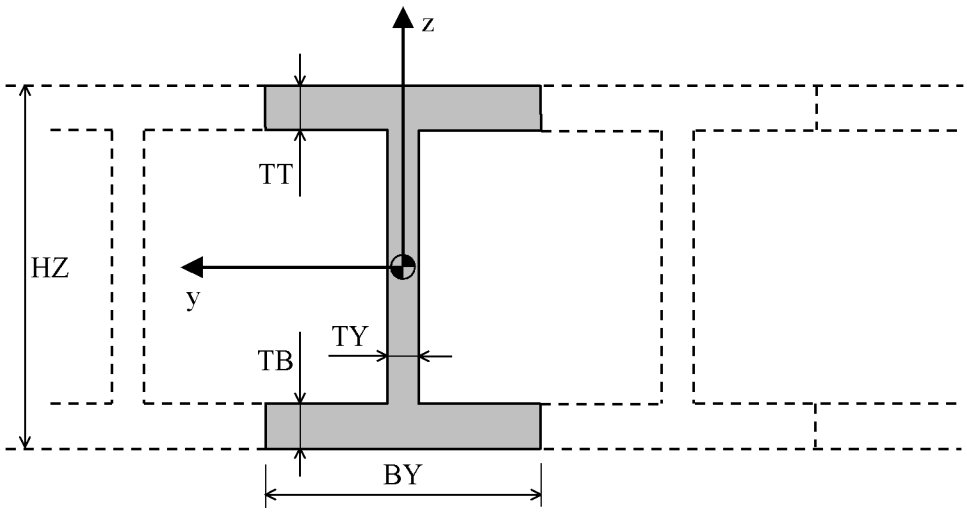


Figure B.4 Double-bottom section

B 1.4.2 Sectional Parameters Computed

The calculation procedure for the double bottom section is the same as for the I section for computation of all parameters except *IX* and *WXMIN*. In the formulae below *IXI* is the *IX* for the I section.

$$IX = IXI + HZ^2 BY/(1/TB + 1/TT)$$

$$WXMIN = IX/\mathrm{MAX}(TT, TY, TB)$$

B 1.5 I (or H) section

B 1.5.1 Sectional Dimensions

HZ	Height
BT	Width of top flange
TT	Thickness of top flange
TY	Thickness of web
BB	Width of bottom flange
TB	Thickness of bottom flange
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

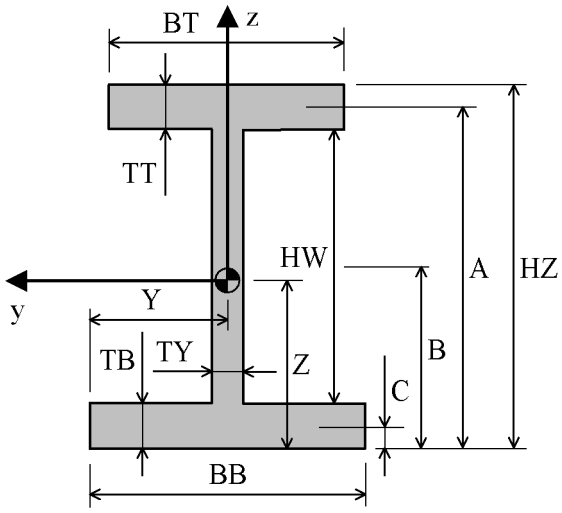


Figure B.5 I (or H) section

B 1.5.2 Sectional Parameters Computed

The expressions below for *IX*, *IY*, *IZ*, *WXMIN*, *WYMIN*, *WZMIN*, *SHARY*, *SHARZ* and SHCENY are taken from Ref. /1/. The expression for *SHCENZ* is taken from Ref. /2/.

$$HW = HZ - TT - TB$$

$$A = TB + HW + TT/2$$

$$B = TB + HW/2$$

$$C = TB/2$$

$$AREA = TY \cdot HW + BT \cdot TT + BB \cdot TB$$

$$Y = \mathrm{MAX}(BB/2, BT/2)$$

$$Z = (BT \cdot TT \cdot A + HW \cdot TY \cdot B + BB \cdot TB \cdot C)/AREA$$

$$TRA = BT \cdot TT^3/12 + BT \cdot TT \cdot (HZ - TT/2 - Z)^2$$

$$TRB = TY \cdot HW^3/12 + TY \cdot HW \cdot (TB + HW/2 - Z)^2$$

$$TRC = BB \cdot TB^3/12 + BB \cdot TB \cdot (TB/2 - Z)^2$$

If $(TT = TY$ and $TT = TB)$ then $\begin{cases} IX = TT^3(HW + BT + BB - 1.2TT)/3 \\ WXMIN = IX/TT \end{cases}$

Else $\begin{cases} IX = 1.30(BT \cdot TT^3 + HW \cdot TY^3 + BB \cdot TB^3)/3 \\ WXMIN = IX/\mathrm{MAX}(TT, TY, TB) \end{cases}$

$$IY = TRA + TRB + TRC$$

$$IZ = (TB \cdot BB^3 + HW \cdot TY^3 + TT \cdot BT^3)/12$$

$$IYZ = 0$$

$$WYMIN = IY/\mathrm{MAX}(HZ - Z, Z)$$

$$WZMIN = 2IZ/\mathrm{MAX}(BB, BT)$$

$$SZ = (TT \cdot BT^2 + TB \cdot BB^2 + HW \cdot TY^2)/8$$

$$SHARY = (IZ/SZ) \cdot (TB + TT) \cdot SFY$$

$$SHARZ = (IY/SY) \cdot TY \cdot SFZ$$

$$SHCENY = 0$$

$$SHCENZ = ((HZ - TT/2) \cdot TT \cdot BT^3 + TB^2 \cdot BB^3/2)/(TT \cdot BT^3 + TB \cdot BB^3) - Z$$

$$CY = BB/2$$

$$CZ = Z$$

B 1.6 L section

B 1.6.1 Sectional Dimensions

HZ	Height
TY	Thickness of web
BY	Width of flange
TZ	Thickness of flange
SFY	Shear factor y-direction
SFZ	Shear factor z-direction
POSWEB	=1 for web location in positive y-direction, otherwise =-1

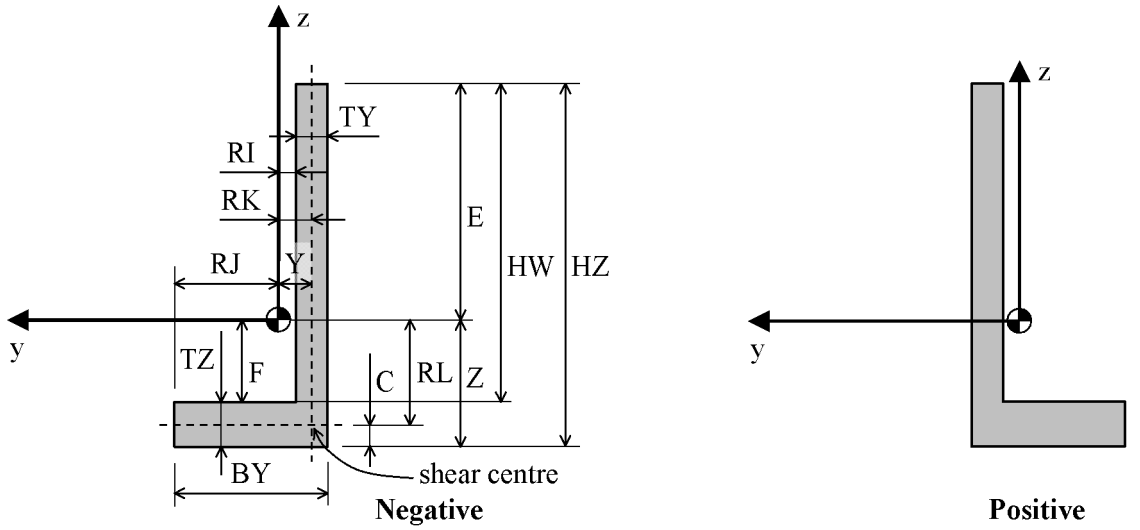


Figure B.6 L section

B 1.6.2 Sectional Parameters Computed

The expressions below for *IX*, *IY*, *IZ*, *WXMIN*, *WYMIN*, *WZMIN*, *SHARY*, *SHARZ* and SHCENY are taken from Ref. /1/. The expression for *SHCENZ* is taken from Ref. /2/.

$$R = 0$$

$$HW = HZ - TZ$$

$$B = TZ - HW/2$$

$$C = TZ/2$$

$$PIQRT = \operatorname{atan} 1.0$$

$$AREA = TY \cdot HW + BY \cdot TZ + (1 - PIQRT) \cdot R^2$$

$$Y = (HW \cdot TY^2 + TZ \cdot BY^2)/(2AREA)$$

$$Z = (HW \cdot B \cdot TY + TZ \cdot BY \cdot C)/AREA$$

$$D = 6R + 2(TY + TZ - \sqrt{4R(2R + TY + TZ) + 2TY \cdot TZ})$$

$$E = HW + TZ - Z$$

$$F = HW - E$$

$$RI = Y - TY$$

$$RJ = BY - Y$$

$$RK = RI + 0.5TY$$

$$RL = Z - C$$

$$\text{If } (TZ \geq TY) \text{ then } \begin{cases} TS = TZ \\ TL = TY \\ BA = BY \\ H = HW \end{cases}$$

$$ALPHA = TL \cdot (0.07 + 0.076R/TS)/TS$$

$$IY = (TY \cdot HW^3 + BY \cdot TZ^3)/12 + HW \cdot TY(B - Z)^2 + BY \cdot TZ(Z - C)^2$$

$$IZ = (HW \cdot TY^3 + TZ \cdot BY^3)/12 + HW \cdot TY \cdot RK^2 + TZ \cdot BY(BY/2 - Y)^2$$

$$IYZ = RL \cdot TZ/2 \cdot (Y^2 - RJ^2) - RK \cdot TY/2 \cdot (E^2 - F^2)$$

$$WXMIN = IX/D$$

$$WYMIN = IY/\operatorname{MAX}(Z, HZ - H)$$

$$WZMIN = IZ/\operatorname{MAX}(Y, RJ)$$

$$SY = E^2 \cdot TY/2$$

$$SZ = RJ^2 \cdot TZ/2$$

$$SHARY = (IZ \cdot TZ/SZ) \cdot SFY$$

$$SHARZ = (IY \cdot TZ/SY) \cdot SFZ$$

If web located in positive y then $\begin{cases} IYZ = -IYZ \\ SHCENY = RK \\ CY = BY - Y \end{cases}$

Else $\begin{cases} SHCENY = -RK \\ CY = Y \end{cases}$

$SHCENZ = -RL$

$CZ = Z$

B 1.7 Pipe section

B 1.7.1 Sectional Dimensions

DY	Outer diameter
T	Thickness of wall
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

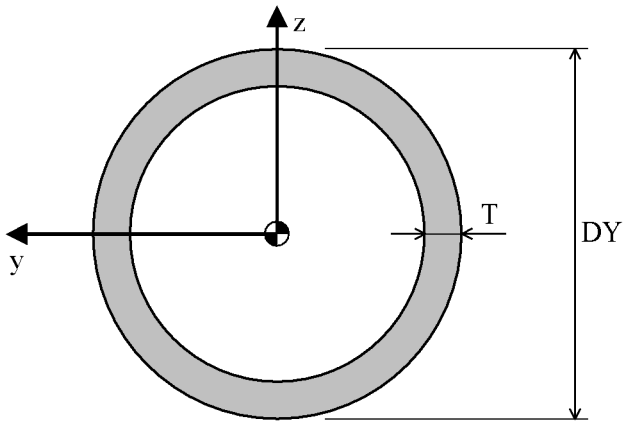


Figure B.7 Pipe section

B 1.7.2 Sectional Parameters Computed

The expressions below for *IX*, *IY*, *IZ*, *WXMIN*, *WYMIN*, *WZMIN*, *SHARY*, *SHARZ*, SHCENY and SHCENZ are taken from Ref. /1/.

$$DI = DY - 2T$$

$$PI = 4\,\mathrm{atan}\,1.0$$

$$AREA = PI \cdot T(DY - T)$$

$$IX = PI(DY^4 - (DY - 2T)^4)/32$$

$$IY = IX/2$$

$$IZ = IY$$

$$IYZ = 0$$

$$WXMIN = 2IX/DY$$

$$WYMIN = 2IY/DY$$

$$WZMIN = 2IZ/DY$$

$$SY = (DY^3 - DI^3)/12$$

$$SZ = SY$$

$$SHARY = (2IZ \cdot T/SY) \cdot SFY$$

$$SHARZ = (2IY \cdot T/SZ) \cdot SFZ$$

$$SHCENY = 0$$

$$SHCENZ = 0$$

B 1.8 Un-symmetrical I section

B 1.8.1 Sectional Dimensions

HZ	Height
BT	Width of top flange
BTA (b1 in Figure 5.44)	Width of part of top flange along positive y-axis
TT	Thickness of top flange
TY	Thickness of web
BB	Width of bottom flange
BBA (b2 in Figure 5.44)	Width of part of bottom flange along positive y-axis
TB	Thickness of bottom flange
SFY	Shear factor y-direction
SFZ	Shear factor z-direction

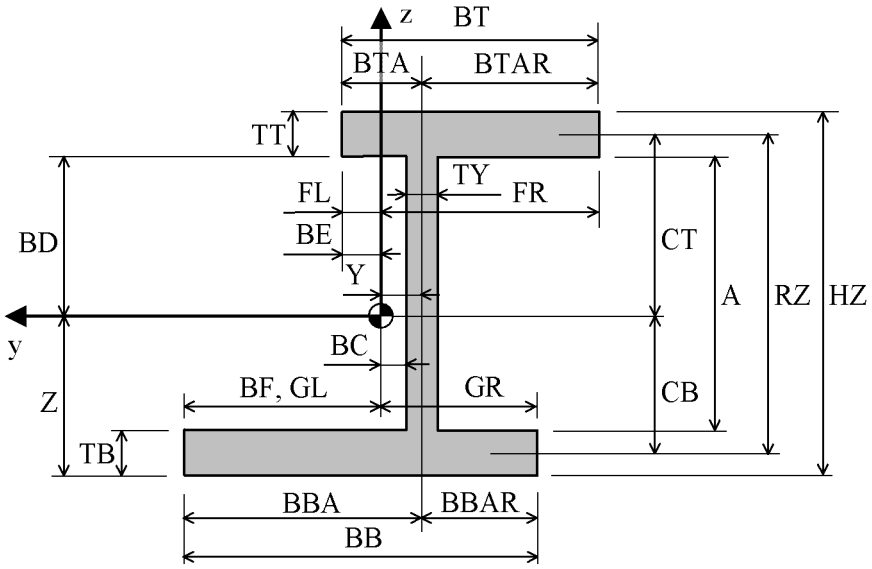


Figure B.8 Un-symmetrical I section

B 1.8.2 Sectional Parameters Computed

The expressions below for *IX*, *IY*, *IZ*, *WXMIN*, *WYMIN*, *WZMIN*, *SHARY* and *SHARZ* are taken from Ref. /1/. The expressions for SHCENY *and SHCENZ* are taken from Ref. /3/.

$$A = HZ - TB - TT$$

$$AREA = BB \cdot TB + BT \cdot TT + A \cdot TY$$

$$Y = (BB \cdot TB(BBA - BB/2) + BT \cdot TT(BTA - BT/2))/AREA$$

$$Z = (BB \cdot TB^2/2 + A \cdot TY(TB + A/2) + BT \cdot TT(HZ - TT/2))/AREA$$

If $(TT = TY)$ and $(TT = TB)$ then $\begin{cases} IX = TT^3(BT + A + BB - 2.6TT)/3 \\ WXMIN = IX/TT \end{cases}$

Else $\begin{cases} IX = 1.1(BT \cdot TT^3 + A \cdot TY^3 + BB \cdot TB^3)/3 \\ WXMIN = IX/\mathrm{MAX}(TT, TY, TB) \end{cases}$

$$IY = (BT \cdot TT^3 + TY \cdot A^3 + BB \cdot TB^3)/12 + BT \cdot TT(HZ - Z - TT/2)^2 + TB \cdot BB(Z - TB/2)^2 + TY \cdot A(TB + A/2 - Z)^2$$

$$IZ = (TT \cdot BT^3 + A \cdot TY^3 + TB \cdot BB^3)/12 + TT \cdot BT(BT/2 - BTA - Y)^2 + BB \cdot TB(BB/2 - BBA + Y)^2 + TY \cdot A \cdot Y^2$$

$$IYZ = -BT \cdot TT(BD + TT/2) \cdot (BT/2 - BTA + Y) - A \cdot TY \cdot Y(TB + A/2 - Z) - BB \cdot TB(TB/2 - Z) \cdot (BB/2 - BBA + Y)$$

$$WYMIN = IY/\mathrm{MAX}(HZ - Z, Z)$$

$$WZMIN = IZ/\mathrm{MAX}(\mathrm{MAX}(BTA, BBA) - Y, \mathrm{MAX}(BT - BTA, BB - BBA) + Y)$$

$$BC = \mathrm{abs}(Y) - TY/2$$

$$BD = HZ - TT - Z$$

$$BE = BTA - Y$$

$$BF = BBA - Y$$

If $(BC < 0)$ then $SZ = TT \cdot BE^2/2 + TB \cdot BF^2/2 + A \cdot BC^2/2$

Else if $(Y < 0)$ then $SZ = (TT \cdot BE^2 + TB \cdot BF^2)/2 + A \cdot TY \cdot BC$

Else $SZ = (TT \cdot BE^2 + TB \cdot BF^2)/2$

$$SY = BT \cdot TT(BD + TT/2) + TY \cdot BD^3/2$$

$$SHARY = (IZ/SZ) \cdot (TB + TT) \cdot SFY$$

$$SHARZ = (IY/SY) \cdot TY \cdot SFZ$$

$$RZ = HZ - (TT + TB)/2$$

If $(Y < 0)$ then $\begin{cases} FL = BTA + \text{abs}(Y) \\ GL = BBA + \text{abs}(Y) \end{cases}$

Else $\begin{cases} FL = BTA - \text{abs}(Y) \\ GL = BBA - \text{abs}(Y) \end{cases}$

$FR = BT - FL$

$GR = BB - GL$

$CT = BD + TT/2$

$CB = Z - TB/2$

$BTAR = BT - BTA$

$BBAR = BB - BBA$

$BG = (BBA^3 + BBAR^3)/3 - (GL \cdot BBA^2 + GR \cdot BBAR^2)$

$SHCENY = -Y + 0.5RZ \cdot TB(IYZ \cdot BG - IZ \cdot CB(BBA^2 - BBAR^2))/(IY \cdot IZ - IYZ^2)$

$BH = (BTA^3 + BTAR^3)/3 - (FL \cdot BTA^2 + FR \cdot BTAR^2)$

$CY = Y + BB - BBA$

$CZ = Z$

B 2 Units

A SESAM analysis is based on a set of consistent units. The units to use must be determined before commencing the modelling and these units must be adhered to throughout the analysis project, i.e. in all SESAM programs employed. The basis for determining a set of consistent units and some examples are given below.

The fundamental equation is:

FORCE = MASS · ACCELERATION

In terms of the fundamental units of mass (M), length (L) and time (T) this equation may be written:

$F = M \cdot L / T^2$

Force, stress, density, etc. are not fundamental units and must be derived in terms of the fundamental units of M, L and T.

The first step in determining a set of consistent units is to select fundamental units. Input values to the programs such as steel density and Young's modulus (input to Prefem) and water density and gravity (input to Wadam) must then be determined in terms of these fundamental units.

Whenever possible it is simplest to use the SI units or multiples of the SI units:

- length in metres (m)
- mass in kilograms (kg)
- time in seconds (s)

A force will then be in Newton (N):

$1\ N = 1\ kg\ m / s^2$

B 2.1 Example

A model has been generated with centimetres (cm) as length unit. We want our output force unit to be tonnes-force (tonnef) and thus need to know which values for Young's modulus and steel density to specify in Prefem and which values for gravity and water density to specify in Wadam.

We first determine what our fundamental units of M, L and T are:

L is in centimetres (cm)

T is chosen to be in seconds (s)

We have already chosen the force unit to be tonnes-force (tonnef) and we know that:

$1\ tonnef = 9810\ kg\ m / s^2 \approx 10000\ kg\ m / s^2$

Insert this in the fundamental equation $F = M \cdot L / T^2$:

$10000\ kg\ m / s^2 = M \cdot cm / s^2$

$10000\ kg \cdot 100\ cm / s^2 = M \cdot cm / s^2$

Hence:

$M = 10000 \cdot 100 \text{ kg} = 10^6 \text{ kg}$

So our fundamental units are:

M in 10^6 kg

L in cm

T in s

The next step is to determine the density, Young's modulus, etc. in terms of our fundamental units.

Steel density, ρ:

Density = Mass / Volume = M / L^3, thus the derived density shall be in 10^6 kg / cm^3

$\rho_{steel} = 7850 \text{ kg} / m^3 = 7.85 \cdot 10^{-9}$ (10^6 kg / cm^3)

Young's modulus, E:

Young's Modulus = Force / Area = $(M \cdot L / T^2) / L^2 = M / (L \cdot T^2)$, thus the derived Young's modulus shall be in 10^6 kg / $(cm \cdot s^2)$.

$E = 2.1 \cdot 10^{11}$ N / $m^2 = 2.1 \cdot 10^{11}$ kg m / $(s^2 \cdot m^2) = 2.1 \cdot 10^{11}$ kg / $(m \cdot s^2)$

Then in our derived units: $E = 2.1 \cdot 10^{11} / 10^6 \cdot 100$ (10^6 kg / cm · s^2) = $2.1 \cdot 10^3$ (10^6 kg / cm · s^2)

Gravity:

Gravity = Acceleration = L / T^2, thus our derived gravity unit shall be cm / s^2.

$g = 9.81 \text{ m} / s^2 = 9.81 \cdot 10^2$ (cm / s^2) = 981 (cm / s^2)

Sea water density:

Density = Mass / Volume = M / L^3, thus the derived density shall be in 10^6 kg / cm^3.

$\rho_{water} = 1025 \text{ kg} / m^3 = 1.025 \cdot 10^{-9}$ (10^6 kg / cm^3)

B 2.2 Consistent Sets of Units

Tables over sets of consistent units are provided below.

Nomenclature:

cm	centimetres
E	Young's modulus
kg	kilograms
kgf	kilograms-force
L	fundamental length symbol
m	metres

mm	millimetres
M	fundamental mass symbol
N	Newtons
s	seconds
t	tonnes
tonnef	tonnes-force
T	fundamental time symbol
rho	density

Table B.1 Examples of consistent units, time unit is second

Length unit L	Mass unit M	Force unit $M\,L\,/\,T^2$	Typical program input values	
			Density of steel (Mass/Volume) $M\,/\,L^3$	Young's modulus for steel (Force/Area) $M\,/\,(\,L\cdot T^2\,)$
m	kg	1 N	$7.85\cdot10^{3}$	$2.10\cdot10^{11}$
m	10^3 kg = 1 t	10^3 N	7.85	$2.10\cdot10^{8}$
cm	kg	10^{-2} N	$7.85\cdot10^{-3}$	$2.10\cdot10^{9}$
cm	10^3 kg = 1 t	10 N ≈ 1kgf	$7.85\cdot10^{-6}$	$2.10\cdot10^{6}$
mm	kg	10^{-3} N	$7.85\cdot10^{-6}$	$2.10\cdot10^{8}$
mm	10^3 kg = 1 t	1 N	$7.85\cdot10^{-9}$	$2.10\cdot10^{5}$
cm	10^2 kg	1 N	$7.85\cdot10^{-5}$	$2.10\cdot10^{7}$
m	10^4 kg	1 tonnef ≈ 10000 N	$7.85\cdot10^{-1}$	$2.10\cdot10^{7}$
cm	10^6 kg	1 tonnef ≈ 10000 N	$7.85\cdot10^{-9}$	$2.10\cdot10^{3}$
mm	10^7 kg	1 tonnef ≈ 10000 N	$7.85\cdot10^{-13}$	2.10
m	10 kg	1 kgf ≈ 10 N	$7.85\cdot10^{2}$	$2.10\cdot10^{10}$
cm	10^3 kg	1 kgf ≈ 10 N	$7.85\cdot10^{-6}$	$2.10\cdot10^{6}$
mm	10^4 kg	1 kgf ≈ 10 N	$7.85\cdot10^{-10}$	$2.10\cdot10^{4}$

REFERENCES

1 W. Beitz, K.H. Küttner:
"Dubbel, Taschenbuch für den Maschinenbau"
17. Auflage (17th ed.)
Springer-Verlag 1990

2 Arne Selberg:
"Stålkonstruksjoner"
Tapir 1972

3 S. Timoshenko:
"Strength of Materials, Part I, Elementary Theory and Problems"
Third Edition 1995
D. Van Nostrand Company Inc.

4 R.M. Jones
"Mechanics of Composite Materials"
Hemisphere Publishing Corporation 1975